DOCUMENT RESUME

ED 120 550 CE 006 944

AUTHOR Proebstle, Donald C.
TITLE Introduction to Geometric Dimensioning and Tolerancing.
INSTITUTION Rutgers, The State Univ., New Brunswick, N.J. Curriculum Lab.
SPONS AGENCY New Jersey State Dept. of Education, Trenton. Div. of Vocational Education.
REPORT NO VT-102-623
PUB DATE Nov 75
NOTE 84p.
AVAILABLE FROM New Jersey Vocational-Technical Curriculum Laboratory, Rutgers-The State University, Building 4103 Kilmer Campus, New Brunswick, New Jersey 08903 ($2.00)

EDRS PRICE MF-$0.83 HC-$4.67 Plus Postage
DESCRIPTORS Assignments; *Drafting; *Engineering Drawing; Engineering Graphics; *Geometric Concepts; High School Curriculum; Instructional Materials; Manuals; Mathematics Materials; Secondary Education; Study Guides; *Supplementary Textbooks; Technical Education; *Vocational Education; Vocational High Schools
IDENTIFIERS *Tolerance (Measurement)

ABSTRACT

Intended for drafting students in engineering drawing classes, the text provides basic definitions and explanations, as well as sample illustrations to be used as guides in dimensioning class projects by this technique. The content is divided into three units plus an appendix. Unit 1 discusses basic concepts, including the importance of the system; unit 2 covers tolerances of form, and unit 3, tolerances of position. The units are subdivided into lessons. Each lesson states an objective, gives related information (if any), discusses the topic with the aid of accompanying illustrations, and concludes with assignments for the student. These assignments are questions based on the material covered in the lesson and also frequently require the student to perform calculations or draw sketches. Appended are conversion and comparison charts to aid the student in the completion of assignments. (RG)

****
* Documents acquired by ERIC include many informal unpublished *
* materials not available from other sources. ERIC makes every effort *
* to obtain the best copy available. Nevertheless, items of marginal *
* reproducibility are often encountered and this affects the quality *
* of the microfiche and hardcopy reproductions ERIC makes available *
* via the ERIC Document Reproduction Service (EDRS). EDRS is not *
* responsible for the quality of the original document. Reproductions *
* supplied by EDRS are the best that can be made from the original. *
****

ED120550

State of New Jersey
Department of Education
Division of Vocational Education

INTRODUCTION TO GEOMETRIC DIMENSIONING AND TOLERANCING

Donald C. Proebstle, Instructor

Donald Springle, Superintendent
Camden County Vocational and Technical Schools
Sicklerville, New Jersey

CE006944

Vocational-Technical
Curriculum Laboratory
Rutgers-The State University
Building 4103 – Kilmer
New Brunswick, New Jersey

U.S. DEPARTMENT OF HEALTH, EDUCATION & WELFARE
NATIONAL INSTITUTE OF EDUCATION

THIS DOCUMENT HAS BEEN REPRODUCED EXACTLY AS RECEIVED FROM THE PERSON OR ORGANIZATION ORIGINATING IT. POINTS OF VIEW OR OPINIONS STATED DO NOT NECESSARILY REPRESENT OFFICIAL NATIONAL INSTITUTE OF EDUCATION POSITION OR POLICY

November 1975

VT 102 623

NEW JERSEY DEPARTMENT OF EDUCATION – FRED G. BURKE, COMMISSIONER

DIVISION OF VOCATIONAL EDUCATION – STEPHEN POLIACIK, ASSISTANT COMMISSIONER

CURRICULUM LABORATORY

RUTGERS – THE STATE UNIVERSITY

BUILDING 4103 – KILMER CAMPUS

NEW BRUNSWICK, NEW JERSEY

Introduction

Geometric Dimensioning and Tolerancing is a much needed outgrowth of the familiar and accepted rectangular-coordinate system of dimensioning. With our rapidly increasing complex technology, the primary medium is still the engineering drawing.

If the engineering drawing completely describes the part or assembly without question, that part or assembly is converted to a product with success. However, in our complex technological society the coordinate dimensioning system falls short of conveying the necessary information to the people who are going to perform the build function.

It is essential that the drawing tell the complete story as to form, fit, and function. The coordinate system, in many ways, is subject to interpretations resulting from the need to build more preciseness and clarity into the engineering drawing. The advanced technique developed in applying engineering-drawing principles is known as geometric dimensioning and tolerancing.

Geometric dimensioning and tolerancing, used in conjunction with the coordinate dimensioning system, will not right all drawing ills, nor does it replace completely all existing drawing techniques. The system should only be used where it can serve an effective purpose.

Industry is looking for people with a knowledge of this subject, because their own manufacturing facilities, as well as other machine shops, may build their products. In most instances this would give rise to questions of interpretation from one manufacturer to the other, and as a result products may not fit together as designed. Having persons familiar with this dimensioning system insures the product manufacturer that the parts specified with geometric dimensioning and tolerancing will cause him fewer problems. Thus the need to at least make machine-drafting students aware of this technique. Also the need for this text.

I would like to stress the complexity of this subject matter. My intention is to give you basic definitions with brief explanations, and many samples which you may use as a guide to dimensioning the actual lessons or projects which your instructor may see fit to have you draw using this technique.

TABLE OF CONTENTS

Page

Unit I
Basic Concepts of Geometric Dimensioning and Tolerancing

Lesson 1 – Why Is Geometric Dimensioning and Tolerancing Needed? 1
Lesson 2 – Identification of Form and Positional Characteristics 3
Lesson 3 – Identification and Understanding of the Modifiers 7
Lesson 4 – Basics, Datums, Feature-Control Symbols 11

Unit II
Tolerances of Form

Lesson 1 – What Are Form Tolerances? 17
Lesson 2 – Flatness Tolerance 19
Lesson 3 – Straightness Tolerance 21
Lesson 4 – Parallelism Tolerance 23
Lesson 5 – Perpendicularity 29
Lesson 6 – Angularity 33
Lesson 7 – Roundness (Circularity) 37
Lesson 8 – Cylindricity 41
Lesson 9 – Profile Tolerancing 43
Lesson 10 – What Is Runout? 49

Unit III
Tolerances of Position

Lesson 1 – Defining True Position 52
Lesson 2 – True Position Theory 55
Lesson 3 – Mating Parts: Floating-Fastener Condition 59
Lesson 4 – Mating Parts: Fixed-Fastener Condition 63
Lesson 5 – Location of Holes by Angular Dimensions 65
Lesson 6 – Concentricity 69
Lesson 7 – Symmetry 71

Appendix 73

Conversion Formula
True-Position Tolerance Zones/Coordinate Tolerance Zones
Conversion Chart
True-Position Tolerance Zones/Coordinate Tolerance Zones
Conversion of Coordinate Measurements to True-Position Location
Comparison Between MIL-STD-8C and USASI Y14.5 – 1966
– Major Differences
Comparision Between MIL-STD-8C, USASI Y14.5, ISO
– Basic, Tolerance, Datums, Misc.
Comparision Between MIL-STD-8C, USASI Y14.5, ISO
– Symbols

UNIT I
BASIC CONCEPTS OF GEOMETRIC DIMENSIONING & TOLERANCING

Lesson 1 **Why Is Geometric Dimensioning and Tolerancing Needed?**

Objective: You will be able to identify the needs for geometric dimensioning and tolerancing.

Related Information:

1.0 There are many different ways to dimension a detail part, and with today's standards, these ways are all correct. As a drafting student, you may have disagreed on occasion with your instructor on the way to dimension a drawing; you might have been as correct as your instructor. It is not the fault of the instructor or yours; it is the fault of the system, which caused both of you to interpret differently the form, fit, or function of the part that was being dimensioned. This same confusion that caused both you and your instructor to disagree also causes confusion in industry. This confusion leads to mistakes and delays. Geometric dimensioning and tolerancing is an outgrowth of industry's and the military agencies' standardizing dimensioning systems to eliminate these mistakes and delays.

2.0 Some other reasons for studying this subject are as follows:

1. It will save your future employer money. It saves money by providing for maximum tolerances that make parts, tools and gages easier to build.

2. Dimensioning and drawing delineation (principle views of part) and interpretation are uniform, thereby reducing controversy and guesswork.

3. It has become the "spoken word" throughout industry, the military, and even internationally on engineering-drawing documentation. Every engineer, technician, and draftsman should have a working knowledge of the subject.

4. It is now the drafting "state of the art."

5. It insures interchangeability of mating parts.

6. Space-age technology, sophisticated computers, and machines are such that the rectangular-coordinate system alone will not sufficiently describe the part.

There are other important reasons, but the most important reason was left to last. Job competition being what it is, knowledge of this subject will provide you with that something extra which you may need to get the job over another individual who has no knowledge of this very important subject.

ASSIGNMENT:

1. State five rules for correct dimensioning procedures.

2. List five reasons for studying geometric dimensioning and tolerancing.

3. What is the meaning of reason number 6 found in the related information section?

4. Explain interchangeability of moving parts.

5. Does your present textbook contain a chapter on geometric dimensioning and tolerancing or true position? If so, look it over briefly. It will probably give you a brief résumé of the subject. Did you understand what it said?

UNIT I

BASIC CONCEPTS OF GEOMETRIC DIMENSIONING & TOLERANCING

Lesson 2 — **Identification of Form and Positional Characteristics**

Objective: You will be able to identify the concepts of geometric dimensioning and tolerancing.

Related Information:

1.0 Geometric dimensioning and tolerancing provides the draftsman with a method of stating almost any necessary dimensional requirement on a drawing not covered by the rectangular coordinate system with which you are now familiar.

Each part in an assembly has a particular function for which it was designed. The rectangular coordinate system does not always provide us with the means of functionally dimensioning the part. Part features in particular are very hard to specify, and this sometimes forces the designer to write lengthy notes. Part function and relationship are the most important aspects of any design. Geometric dimensioning and tolerancing provide us with the means to specify these relationships.

2.0 Geometric dimensioning and tolerancing should be used when part features are critical, when part interchangeability is a must, and when standard interpretation or tolerances are not applicable.

3.0 Form tolerances and positional tolerances are the basic controls of the system, and we use these controls to totally dimension a part.

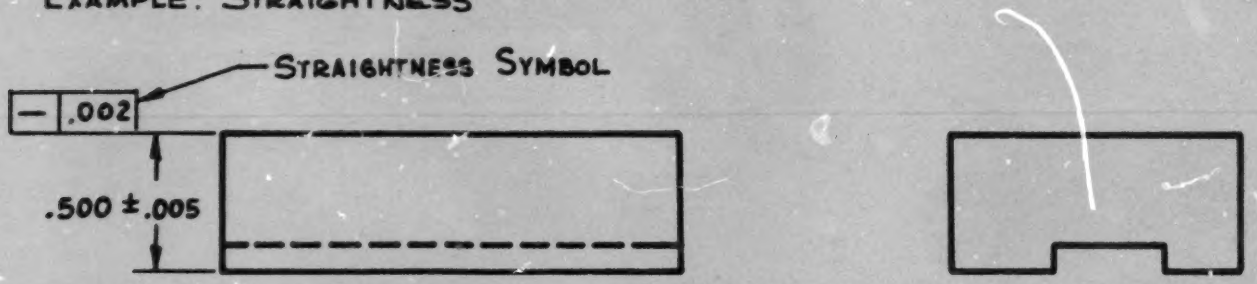
3.1 Form tolerances state how far *features* are permitted to vary from the desired shape specified by the drawing. In Fig. 1 (page 4), you will notice a brief use of form tolerance and see how it differs from the old method. Notice the use of symbols and the brief interpretations and how confusion can be eliminated.

3.2 Expressions of these tolerances are flatness, straightness, parallelism, perpendicularity, angularity, roundness, cylindricity, runout, profile of any surface, and profile of any line. The form characteristics are shown in Fig. 2. It would be well for you at this time to look at the form characteristics and commit some of them to memory.

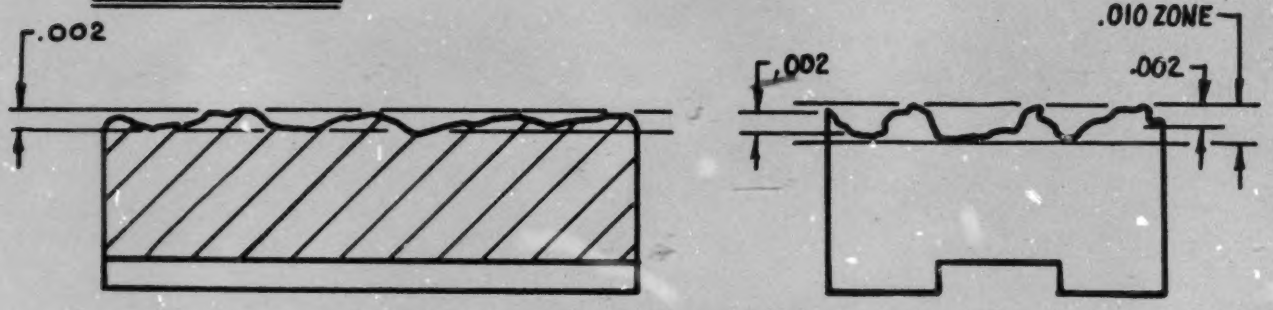
3.3 Positional tolerances show the relationship of center lines, axes, and center planes and indicate the locational relationship of the feature (∥ , ⊥ , ∠,etc.) which we are trying to control. The positional characteristics also are shown in Fig. 2.

EXAMPLE OF GEOMETRIC FORM TOLERANCE

EXAMPLE: STRAIGHTNESS

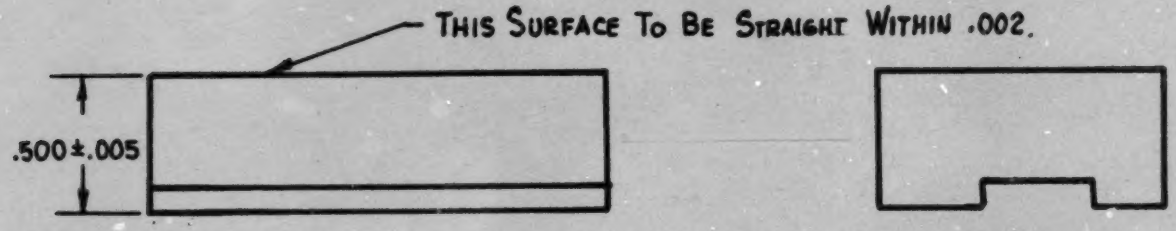


INTERPRETATION



NOTE:
THE SURFACE DEVOTED BY THE STRAIGHTNESS SYMBOL MUST LIE WITHIN .010 ZONE.

EXAMPLE OF RECTANGULAR COORDINATE SYSTEM



NOTE:
THE SURFACE TO BE STRAIGHT WITHIN .002 COULD LIE WITHIN OR OUT OF THE .010 ZONE, THUS LEADING TO CONFUSION.

Figure 1

GEOMETRIC CHARACTERISTICS

▱	FLATNESS	FORM CHARACTERISTICS
—	STRAIGHTNESS	
∠	ANGULARITY	
⊥	PERPENDICULARITY (SQUARENESS, NORMALITY)	
∥	PARALLELISM	
○	ROUNDNESS (CIRCULARITY)	
⌭	CYLINDRICITY	
⌓	PROFILE OF ANY SURFACE	
⌒	PROFILE OF ANY LINE	
↗	RUNOUT	
⌖	TRUE POSITION	POSITION CHARACTERISTICS
◉	CONCENTRICITY	
⌯	SYMMETRY	

MODIFIERS:

- Ⓜ MAXIMUM MATERIAL CONDITION
- Ⓢ REGARDLESS OF FEATURE SIZE

Figure 2

ASSIGNMENT:

1. List the form characteristics and draw the appropriate symbols.
2. List the position characteristics and draw the appropriate symbols.
3. What is geometric dimensioning and tolerancing?
4. What is the difference between form and feature, or are they the same?
5. What is a form tolerance?
6. What is a positional tolerance?

UNIT I

BASIC CONCEPTS OF GEOMETRIC DIMENSIONING & TOLERANCING

Lesson 3 — **Identification and Understanding of the Modifiers**

Objectives: You will be able to state the advantages of symbols over notes.
You will be able to identify the Maximum Material Condition concept.
You will be able to identify the Regardless of Feature Size concept.

Related Information:

1.0 Look at the illustration in Fig. 3. The advantages are evident.

Symbols are more quickly drawn and because of their compactness can be applied at or near the place on the drawing to which they apply.

By combinations of these symbols, plus box tolerances, a requirement may be clearly stated which would otherwise require a long note. These notes usually are inconsistently written and raise questions of interpretation which lead to confusion. They take up drawing space, make for a cluttered drawing, and, most important, necessitate visually skipping back and forth from the main content of the drawing to the notes and vice versa. Notes require memorization before the drawing can be properly interpreted. Symbols are in view and are instantly interpretable at the place of application on the drawing. Note too, how symbols lend more order and preciseness to the drawing's appearance.

2.0 The **maximum material condition**, usually abbreviated to MMC, means that a feature of a finished product contains the maximum amount of material permitted by the toleranced size dimensions shown for that feature. Therefore with MMC, holes, slots, or other *internal* features are at minimum size, and shafts, protrusions, and other *external* features are at their maximum size. See Fig. 4 for a better understanding of thir.

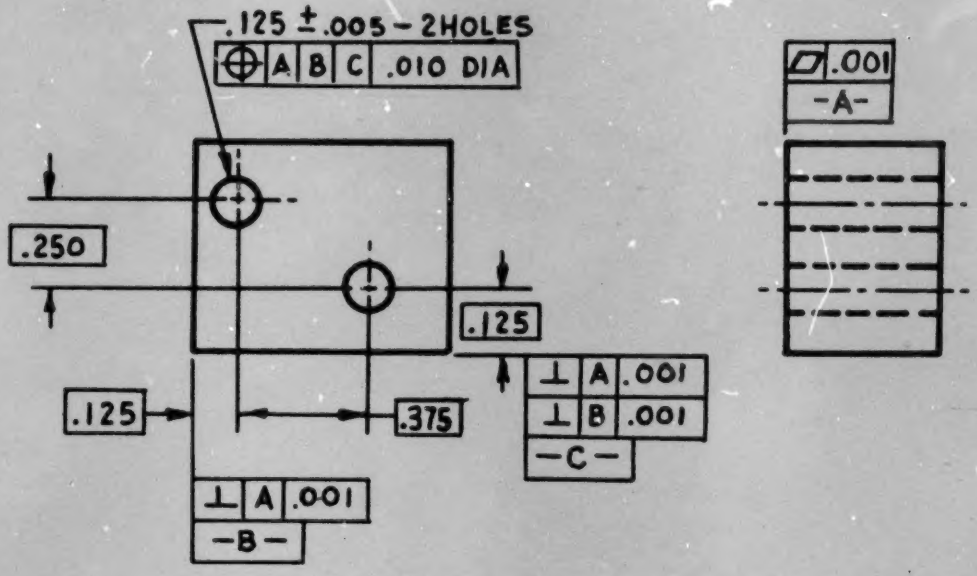
2.1 This is one of the fundamental and most important principles of geometric dimensioning and tolerancing, and a thorough understanding is very important.

2.2 Generally, the use of MMC permits greater possible tolerance as a part *feature* varies from its calculated MMC limits.

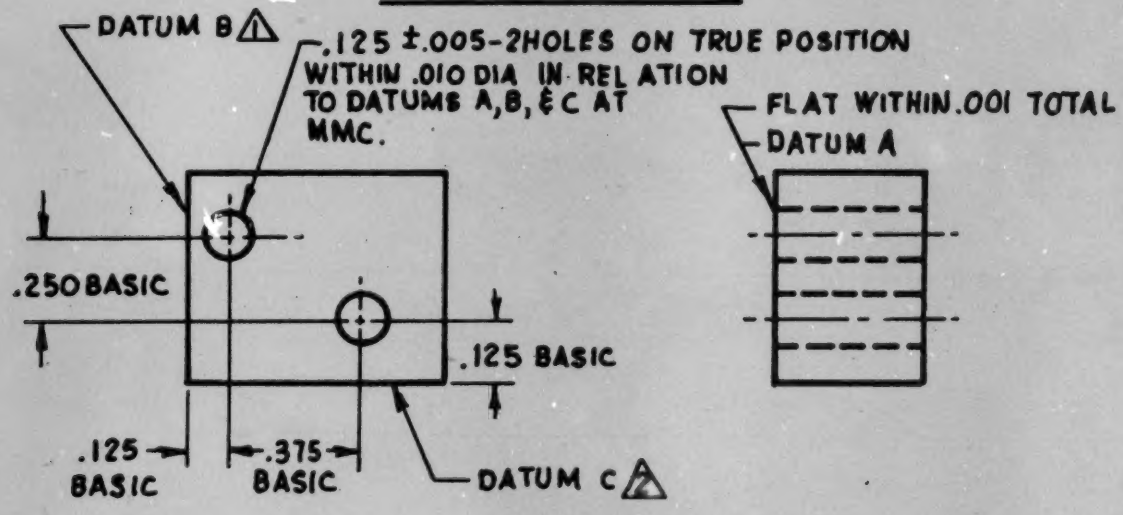
3.0 **Regardless of Feature Size** – Symbol (S), abbreviation RFS.

3.1 Definition: The tolerance of *form or position* of a feature must be met regardless of where the feature lies within its size tolerance.

USING SYMBOLS



USING NOTES


DATUM B ⚠1
.125 ±.005-2HOLES ON TRUE POSITION WITHIN .010 DIA IN RELATION TO DATUMS A, B, & C AT MMC.
FLAT WITHIN .001 TOTAL
DATUM A
.250 BASIC
.125 BASIC
.125 BASIC
.375 BASIC
DATUM C ⚠2

⚠2 SURFACE C PERPENDICULAR TO DATUM A WITHIN .001 & WITH DATUM B WITHIN .001.

⚠1 SURFACE B PERPENDICULAR TO DATUM A WITHIN .001

Figure 3

SYMBOL

Ⓜ

ABREVIATION

MMC

MAXIMUM MATERIAL CONDITION

DEFINITION:

THAT CONDITION OF A PART FEATURE WHEREIN IT CONTAINS THE MAXIMUM AMOUNT OF MATERIAL, eg. MINIMUM HOLE SIZE & MAXIMUM SHAFT SIZE.

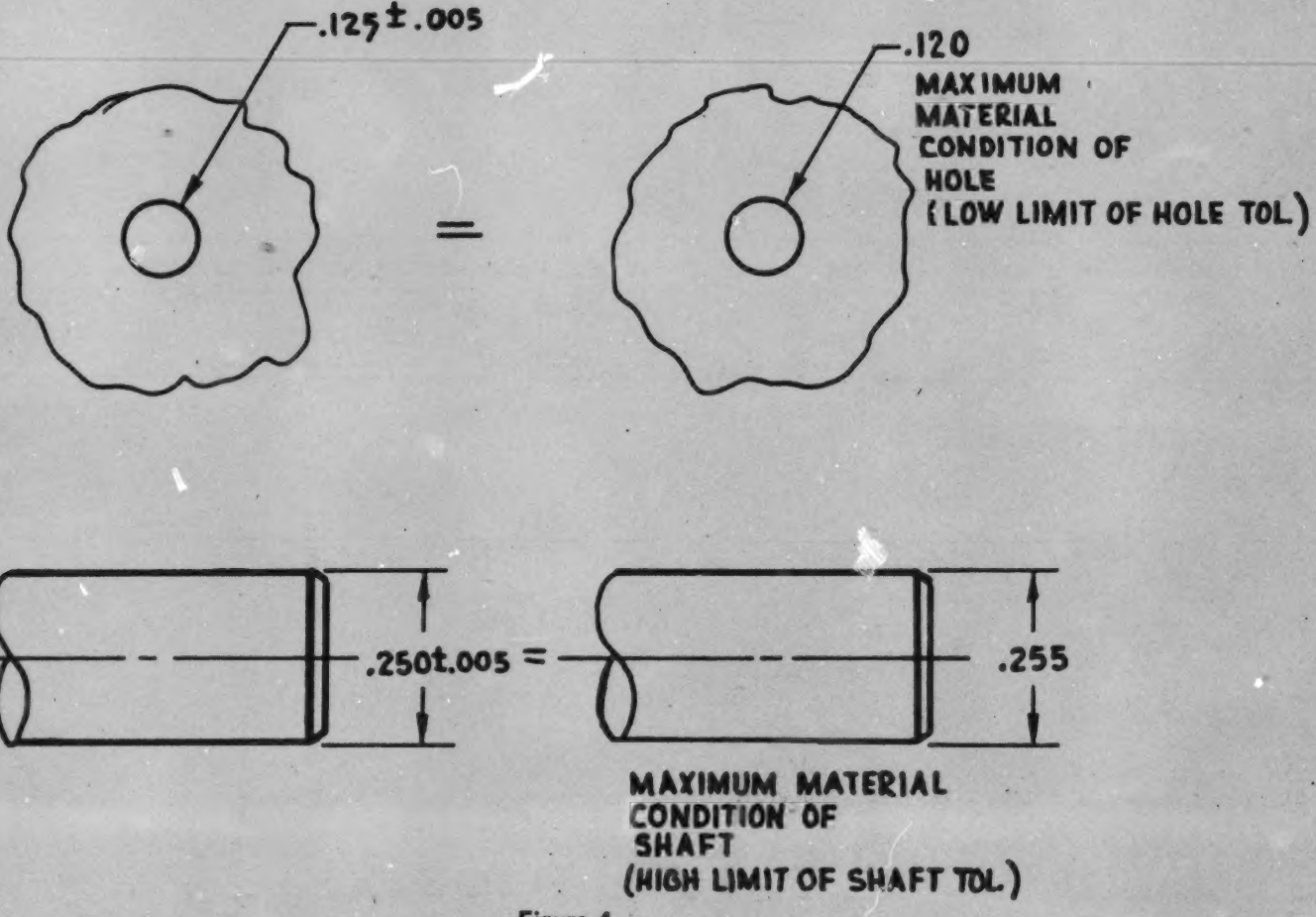
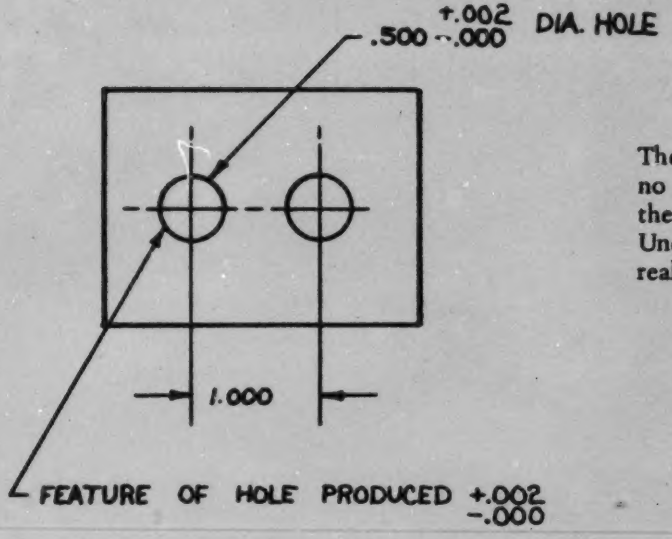


Figure 4

3.2 RFS is another important principle which must be well understood. It differs from MMC in that it allows no additional or form tolerance, no matter to which size the related features are produced. Think of it as being the type of dimensioning you have used in the past. See example below.



The hole is the form tolerance and no additional tolerance is gained on the 1.000 location to "A" hole. Under MMC the + .002 tolerance is realized.

3.3 At this point you should be able to see that additional tolerance is available when the feature size departs from maximum material condition. When using Regardless of Feature Size (S), the tolerance remains constant, and no additional tolerances are realized.

4.0 Special Note: The information presented in the text suggests that either (M) or (S) must be used. In the United States, the implied method is used. That is, if no modifier is present, (M) is the implied positional tolerance.

ASSIGNMENT:

1. List five reasons why symbols are more advantageous than notes.
2. Explain in detail the concept of maximum material condition.
3. Explain the concept of Regardless of Feature Size.
4. What is the symbol for MMS?
5. What is the symbol for RFS?

UNIT I

BASIC CONCEPTS OF GEOMETRIC DIMENSIONING & TOLERANCING

Lesson 4 — **Basics, Datums, Feature-Control Symbols**

Objective: You will be able to state the meaning of the terms "basic," "datum," and "feature control," and show how they are specified on drawings.

Related Information:

1.0 Basic

1.1 Definition: A dimension specified on a drawing as basic is a theoretical value used to describe the exact size, shape, or location of a feature. It is used as the basis from which permissible variations are established by tolerances on other dimensions or in notes.

1.2 Use of a basic dimension, which is a theoretical and exact value, requires also a *tolerance* stating the permissible variation from this exact value (most often relative to a position or form requirement, e.g., hole to hole). A basic dimension states only *half* the requirement. To complete it, a tolerance must be associated with the features involved in the basic dimension.

1.3 A basic dimension is identified on the drawing by the word "basic" (or the accepted abbreviation BSC) adjacent to, or below, the dimension, or by a general note on the drawing. See (1) on Fig. 5.

1.4 The preferred method of stating an "exact" value replacing basic, BSC, TP, etc. is the international (ISO) method recommended by USASI Y14.S. According to this method, the exact value is enclosed in a frame or box. See (2) on Fig. 5. The meaning is the same as that of basic. This is the most acceptable method used in the country.

2.0 Datums

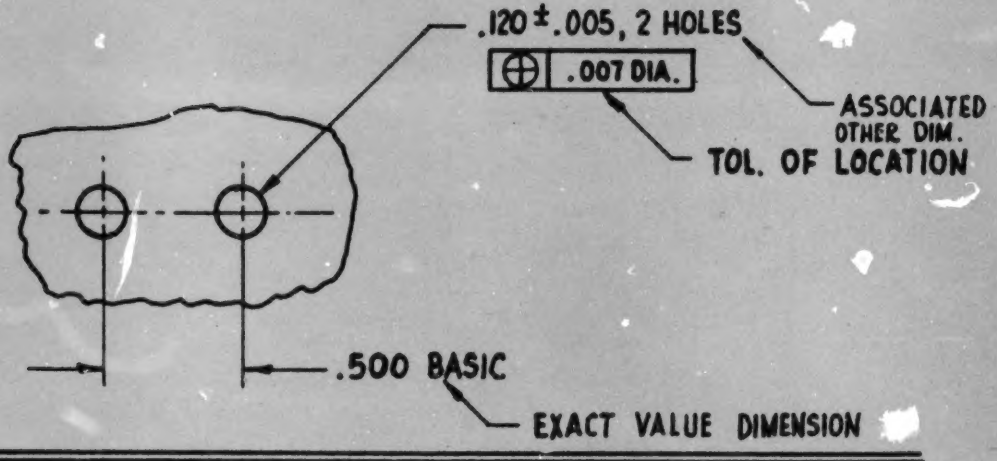
2.1 Definition: Datums are points, lines, planes, cylinders, etc. which are assumed to be exact for purposes of computation or reference and from which the location or form of features of a part may be established. Datums are established by, or relative to, actual part features or surfaces.

2.2 Datum surfaces and datum features are actual part surfaces or features used to establish datums. The datums include all the surface or feature inaccuracies.

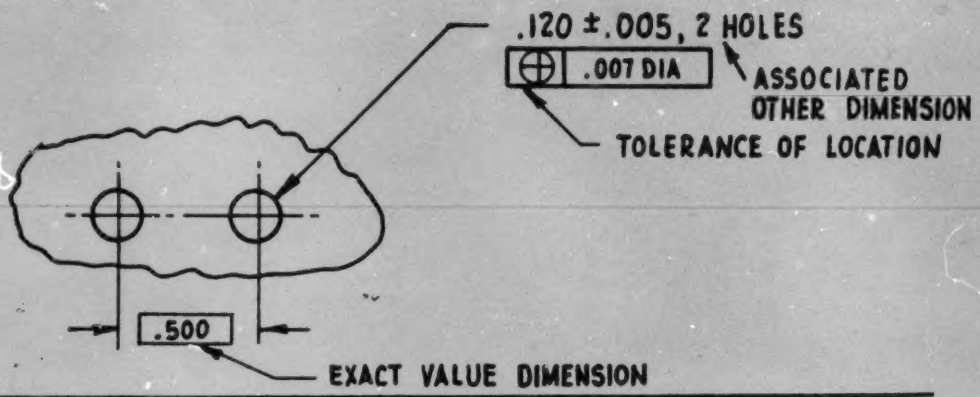
2.3 To identify a feature as a datum, use the datum identification symbol. See (3) on Fig. 5.

DATUM ILLUSTRATIONS

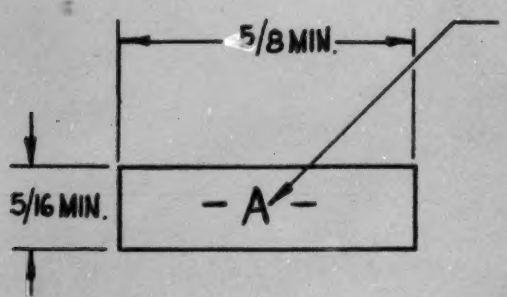
(1)



(2)



(3)



IDENTIFYING LETTER/S (I, O, Q NOT TO BE USED). IF SINGLE LETTERS EXHAUSTED, USE DOUBLE LETTERS. eg., AA, BB, CC, ETC.

NOTE:

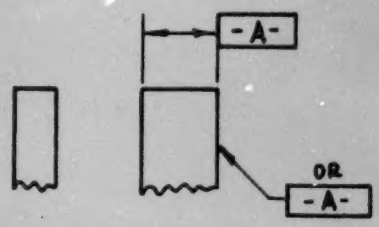
EACH DATUM REQUIRING IDENTIFICATION IS ASSIGNED A DIFFERENT REFERENCE LETTER.

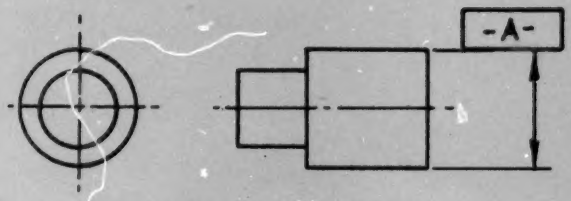
Figure 5

DATUM APPLICATIONS

APPLICATION TO PLANE SURFACES (OR CYLINDERS)

(1)

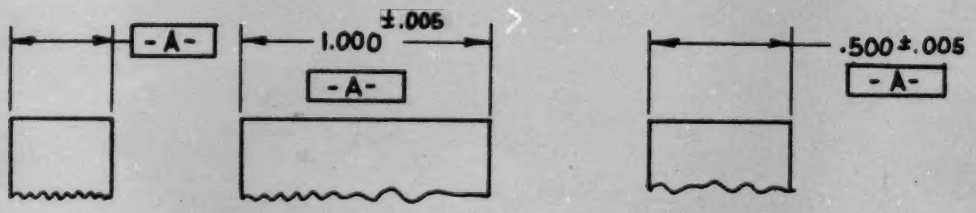




DATUM SYMBOL MAY BE ON EXTENSION LINE IF FEATURE IS A CYLINDER.

(2)

APPLICATION TO FEATURE SIZE



(3)

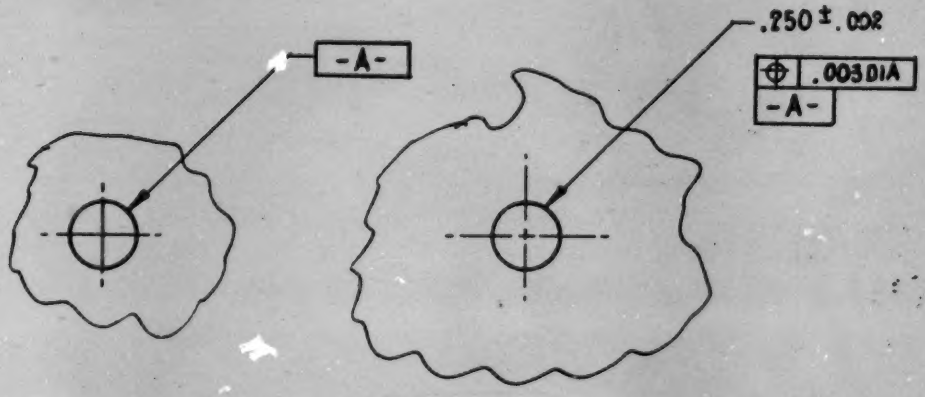
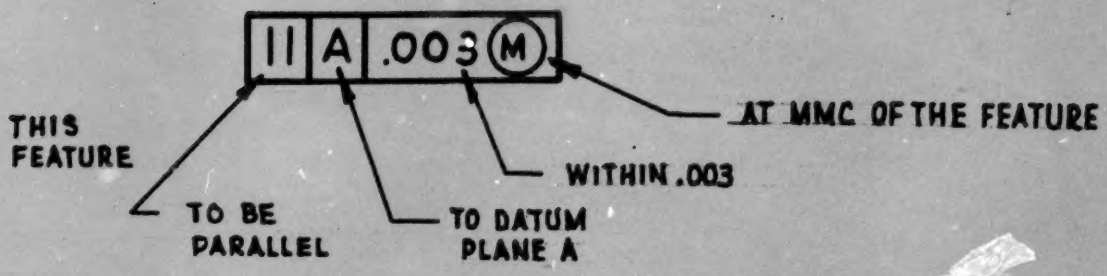
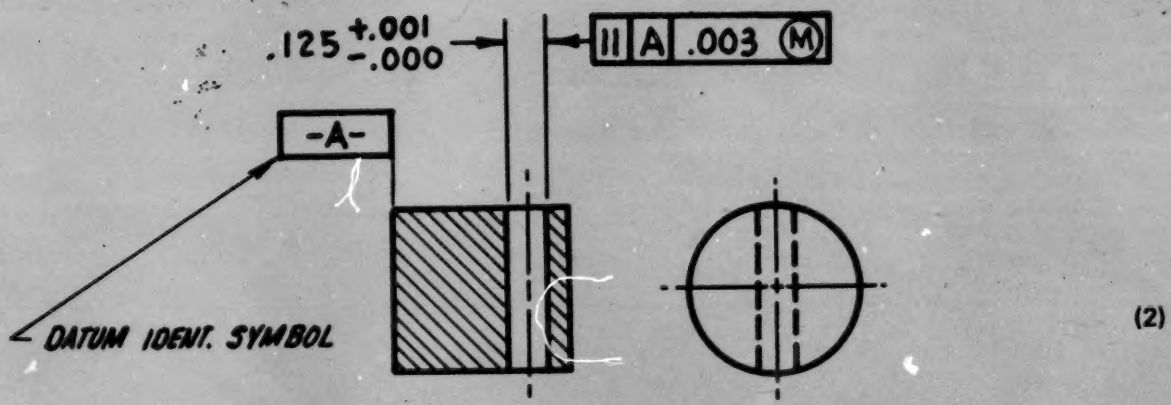


Figure 6
13

FEATURE CONTROL SYMBOL



(1)



(2)

THE FEATURE CONTROL SYMBOLS ARE ASSOCIATED WITH THE FEATURE(S) (IE. HOLE, BOSSES, ETC) BEING TOLERANCED BY ONE OF THE FOLLOWING METHODS.

1.- RUNNING A LEADER LINE FROM THE SYMBOL TO THE FEATURE.
2.- ATTACHING A SIDE, END, OR CORNER OF THE SYMBOL BOX TO AN EXTENSION LINE FROM THE FEATURE. **(USED ON MOST FORM TOLERANCES)**
3.- ATTACHING A SIDE OR END OF THE SYMBOL BOX TO THE **DIMENSION LINE** PERTAINING TO THE FEATURE WHEN IT IS CYLINDRICAL.
4.- PLACING THE SYMBOL BOX BELOW OR CLOSELY ADJACENT TO THE DIMENSION OR NOTE PERTAINING TO THE FEATURE.

Figure 7

2.4 The following conventions will be used concerning datum placement.

1. If the datum identification symbol is shown on the *extension line* of a feature, the datum is *for that feature only* (See (1) and (2) on Fig. 6).

2. If the datum identification symbol is shown on a *dimension line*, below or adjacent to a *dimension* with a leader or note or with a feature-control symbol, then it applies to the *entire dimension or feature* with which it is associated. (See (3) Fig. 6).

3.0 Feature-Control Identification

3.1 Features with associated datums and tolerances are combined into a symbol which consists of a frame containing the geometric characteristic symbol, datum reference, tolerance, and modifier (MMC & RFS). Remember – holes, cylinders, and bosses are features. On Fig. 7, (1) shows a typical feature-control symbol, and (2) shows how the feature-control symbol is used on a part drawing.

Note: Rules for placement of the feature-control symbol are given below (2). You will observe that the placement of the symbol in (2) illustrates rule #3 below.

3.2 Figure 8 (1) and Fig. 8(3) "B" show examples of rule 1. Fig. 8(2) shows examples of rule 2. Part "A" of Fig. 8(3) shows an example of rule 4.

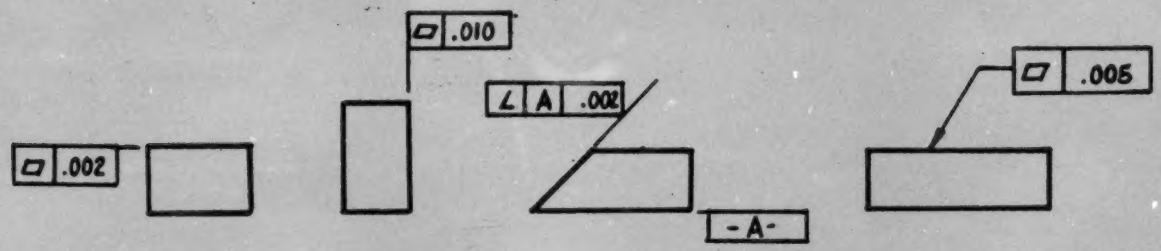
3.3 When a feature is a datum (See Fig. 8(4)) and is also controlled by a positional or form tolerance, the feature-control symbol and the datum identifying symbol are combined as shown.

ASSIGNMENT:

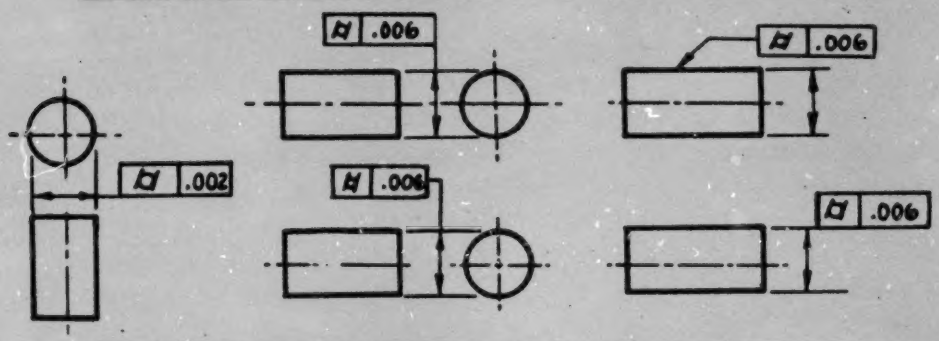
1. What is the meaning of the term "basic"?
2. How is a basic dimension shown on a drawing?
3. What must accompany a basic dimension in order to complete the dimension?
4. What is the meaning of "datum"?
5. What are some part features used to specify datums?
6. Name two conventions that determine datum placement.
7. What are features?
8. Sketch some applications of the specification of feature control. Use examples other than those shown.

PLACEMENT OF FEATURE CONTROL SYMBOL

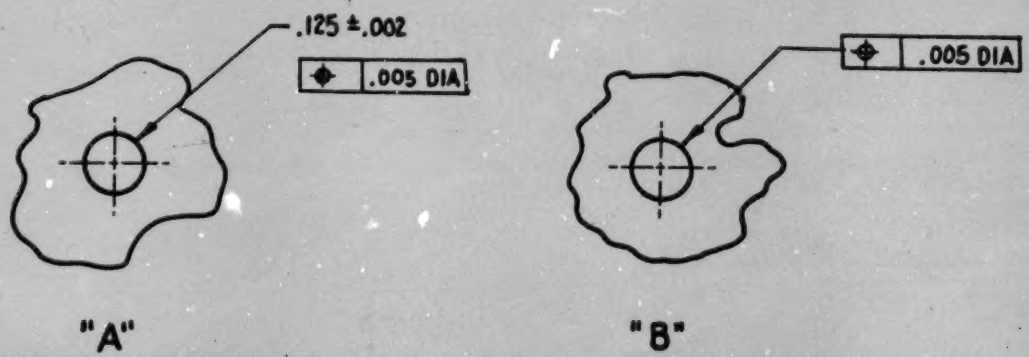
(1)



(2)



(3)



COMBINED FEATURE CONTROL SYMBOL & DATUM IDENTIFYING SYMBOL.

(4)

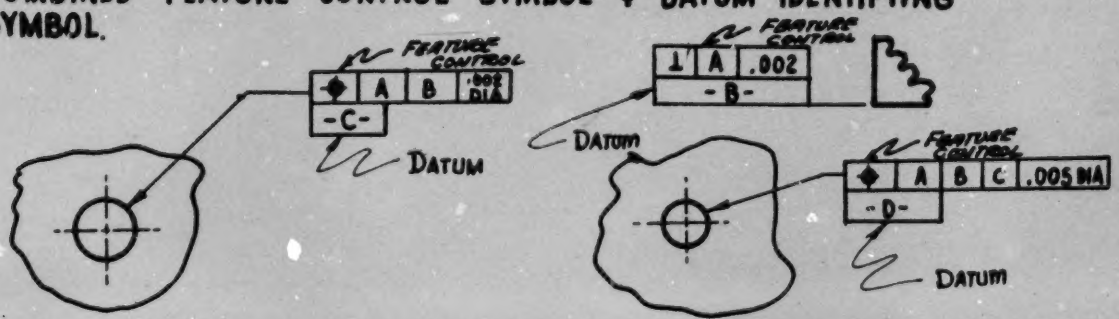


Figure 8
16

UNIT II
TOLERANCES OF FORM

Lesson 1 — **What are form tolerances?**

Objective: You will be able to state what form tolerances are and be able to reproduce the symbols.

Related Information :

1.0 Tolerances of Form

1.1 In lesson 2 of Unit 1 you were given a brief view of what tolerances of form are. The lessons in Unit 2 will be devoted to a detailed study of individual form tolerances and their application to specifications on detail-part drawings.

1.2 Form tolerances state how far actual surfaces or features are permitted to vary from the form specified by the drawing. The forms most often used are flatness, straightness, parallelism, perpendicularity, angularity, roundness, cylindricity, run-out, profile of any surface, and profile of any line.

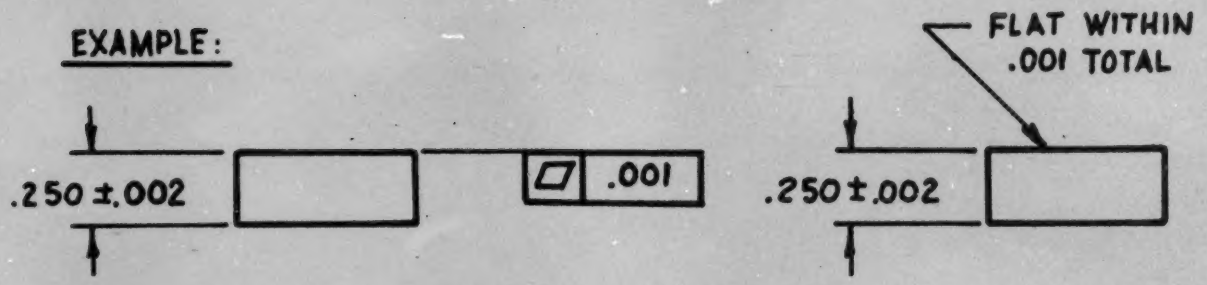
1.3 Symbolic notations of form tolerances will be used and are recommended in actual practice. Review the symbols given in Fig. 2.

1.4 An important point to remember when using form tolerances is that in no case must the size tolerance be violated.

ASSIGNMENT:

1. How many form tolerances are there?
2. Sketch all the symbols for the form tolerances.
3. Features are controlled by form tolerance and suggest _____ of surface error.

FLATNESS ▱



INTERPRETATION OF ABOVE

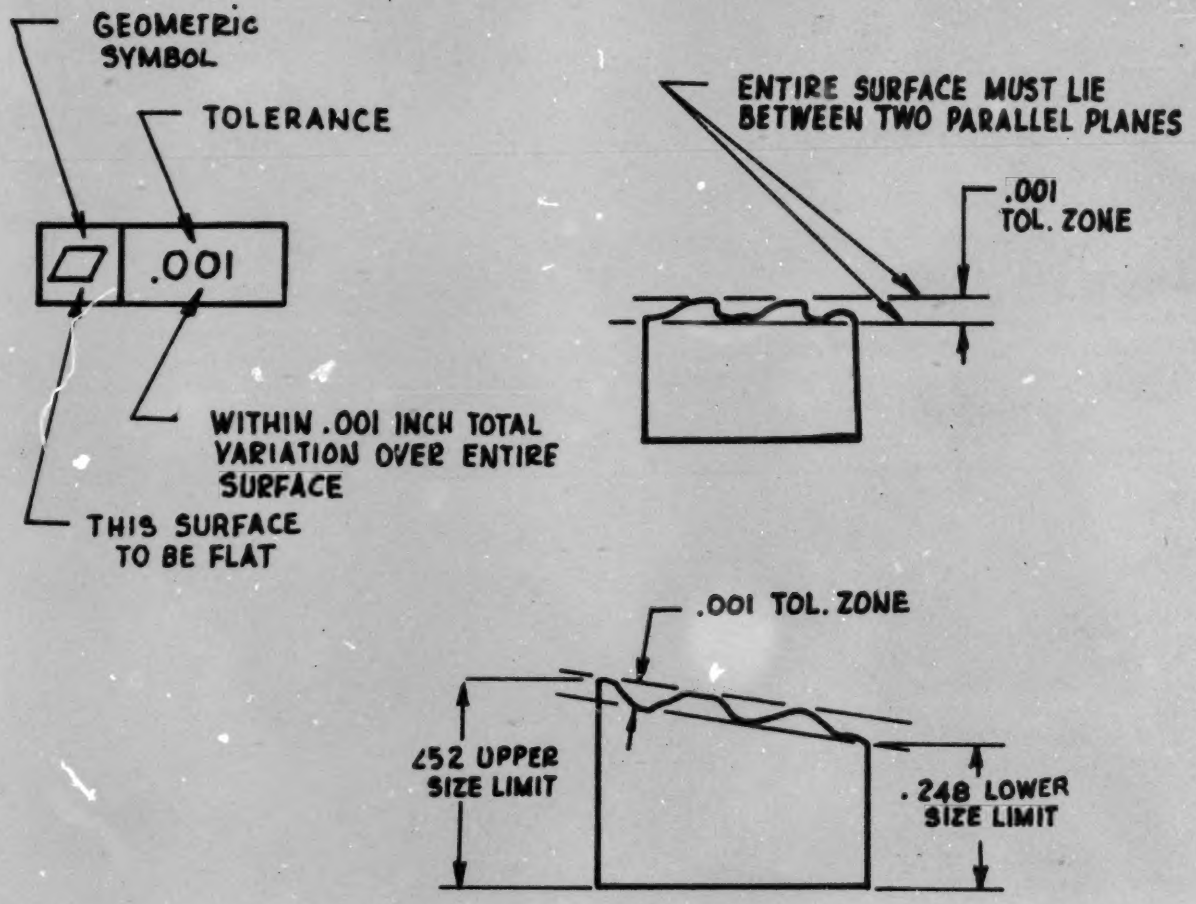


Figure 9

UNIT II
TOLERANCES OF FORM

Lesson 2 — **Flatness Tolerance**

Objective: To identify and specify the flatness tolerance.

Related Information:

1.0 Flatness Tolerance:

1.1 Definition: Flatness is the condition of a surface in which all surface elements are in one plane.

1.2 A flatness tolerance is a form control over all the elements of a surface when compared to a perfect flat surface (which is a plane). The flatness tolerance specifies a *tolerance zone* bounded by two parallel planes, within which the entire surface must lie.

1.3 Fig. 9 shows how the symbol is applied. The long-form notation appears to the right of the symbolic notation.

The symbol is interpreted to read, "This surface shall be flat within .001 total variation over the entire surface." Note that the .001 is the *total* tolerance zone. To be acceptable, the entire actual surface must fall between the parallel extremities of the .001 tolerance zone.

1.4 No datum references are required, since flatness control is a relationship of a feature to itself. The bottom surface of the .250 block has a flat face: the feature is the flat face on the bottom. The top of the block must be flat to the bottom of the block within the .001 tolerance.

1.5 Do not modify a flatness tolerance with an MMC application.

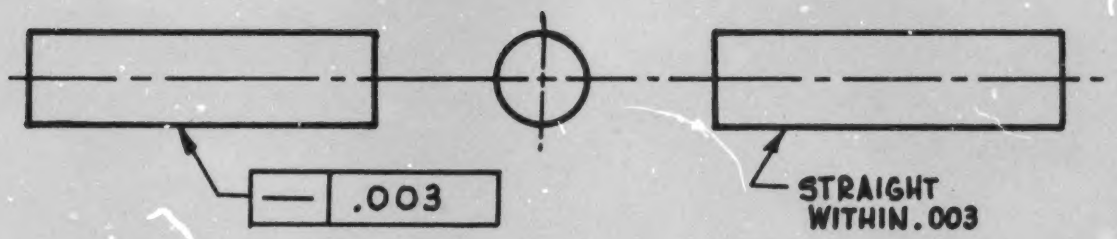
1.6 When a flatness tolerance is specified, the flatness-tolerance zone must be contained within the size-tolerance limits. *Do not add it* to the size tolerance. For example, the dimension here is .250 ± .002, or .250 + .002 = .252 *total*, not .252 + .001 or .253. See the interpretation in Fig. 9.

ASSIGNMENT:

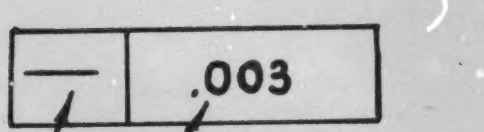
1. What is the symbol for flatness tolerance?
2. A flatness tolerance is .010 and the part dimension is .250 ± .005. What is the maximum dimension? What is the minimum dimension?
3. Sketch an application of flatness with all tolerances specified.

STRAIGHTNESS

EXAMPLE

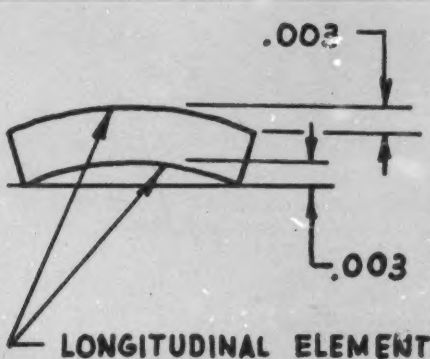


INTERPRETATION OF THE ABOVE



WITHIN .003 INCH REGARDLESS OF FEATURE SIZE.

EACH LONGITUDINAL ELEMENT OF THIS SURFACE SHALL BE STRAIGHT.



LONGITUDINAL ELEMENTS

REGARDLESS OF THE ACTUAL SIZE OF THE FEATURE, ANY LONGITUDINAL ELEMENT OF ITS SURFACE MUST LIE BETWEEN TWO PARALLEL STRAIGHT LINES (.003 APART) WHERE THE TWO LINES AND THE NOMINAL AXIS OF THE FEATURE SHARE A COMMON PLANE..

Figure 10

UNIT II
TOLERANCES OF FORM

Lesson 3 — **Straightness Tolerance**

Objective: You will be able to identify and specify the straightness tolerance.

Related Information:

1.0 Straightness Tolerance

1.1 Definition: Straightness is a condition where an element of a surface is a straight line.

1.2 Straightness tolerance is a form control of longitudinal elements (lengthwise dimension) of cylindrical or conical (cone) surfaces.

1.3 A straightness tolerance specifies a tolerance zone of uniform width along a straight line within which all points of the considered line must lie. This tolerance zone is a fixed width bounded by two straight lines.

1.4 In specifying a straightness tolerance, the symbol is applied to the surface under consideration, and applied in a view in which the surface elements to be controlled are represented by a straight line.

1.5 A straightness tolerance of form is shown in Fig. 10. The symbol could be read "Each longitudinal element of this surface shall be straight within .003 inch."

The illustration at the far right of the figure shows two longitudinal elements, which may only vary by .003. If a cylinder was being inspected, this .003 could occur anywhere along its entire length.

1.6 No datum reference is needed when straightness tolerancing is used. Straightness is a form control of a single element as it is compared to a simulated perfect counterpart of itself. Straightness tolerance cannot be modified by MMC, because it is a form tolerance of a single element.

1.7 Straightness tolerance is *not additive* to size tolerance and must be contained within the limits of size. An example of this would be: A block .250 ± .005 deep has a straightness tolerance of .003. The high limit would be .255 with a straightness tolerance zone of .003, and a .245 low limit, also with a straightness tolerance zone of .003.

ASSIGNMENT:

1. Define the term "straightness".
2. Sketch an application of straightness with all tolerances specified.
3. How does flatness differ from straightness?
4. Can straightness tolerance be used to control the surface of a cylinder?

UNIT II
TOLERANCES OF FORM

Lesson 4 — **Parallelism Tolerance**

Objective: You will be able to identify and specify the parallelism tolerance.

Related Information:

1.0 Parallelism

1.1 Definition: Parallelism is the condition of a surface, line, or axis in which all points are equidistant from another surface, line, or axis.

1.2 A parallelism tolerance specifies either of the following:

a. A tolerance zone between two parallel planes which are in turn parallel to a specified datum plane. The specified feature must then lie between the two zones. See Fig. 11, 3A and 3B.

b. A tolerance zone which is cylindrical and parallel to a datum feature axis, within which the axis of the feature must lie. See Fig. 11, 3C.

1.3 The illustration in Fig. 11(1) shows that the bottom surface has been selected as the datum surface and the top surface must be parallel to the datum surface "A" within .002. The equivalent note (old method) is shown to the right.

Fig. 11(2) illustrates the tolerance zone and the way in which the surface must fall within the tolerance zone to be acceptable. Note that the tolerance zone is established parallel to datum plane A. Note also that the parallelism tolerance, when applied to a plane surface, controls flatness if a flatness tolerance is not specified.

1.4 Specification of parallelism is best explained by showing applications, which are presented in Figs. 12 and 13. Note that when the datum is a plane, the tolerance zone is defined by two planes, and when the datum is a cylinder, the tolerance zone is defined as a cylinder.

1.5 Note also how the MMC applications give the possibility of greater tolerance. The RFS examples should also be analyzed until you understand what RFS means. The examples are self explanatory.

ASSIGNMENT:

1. Define the term "parallelism."

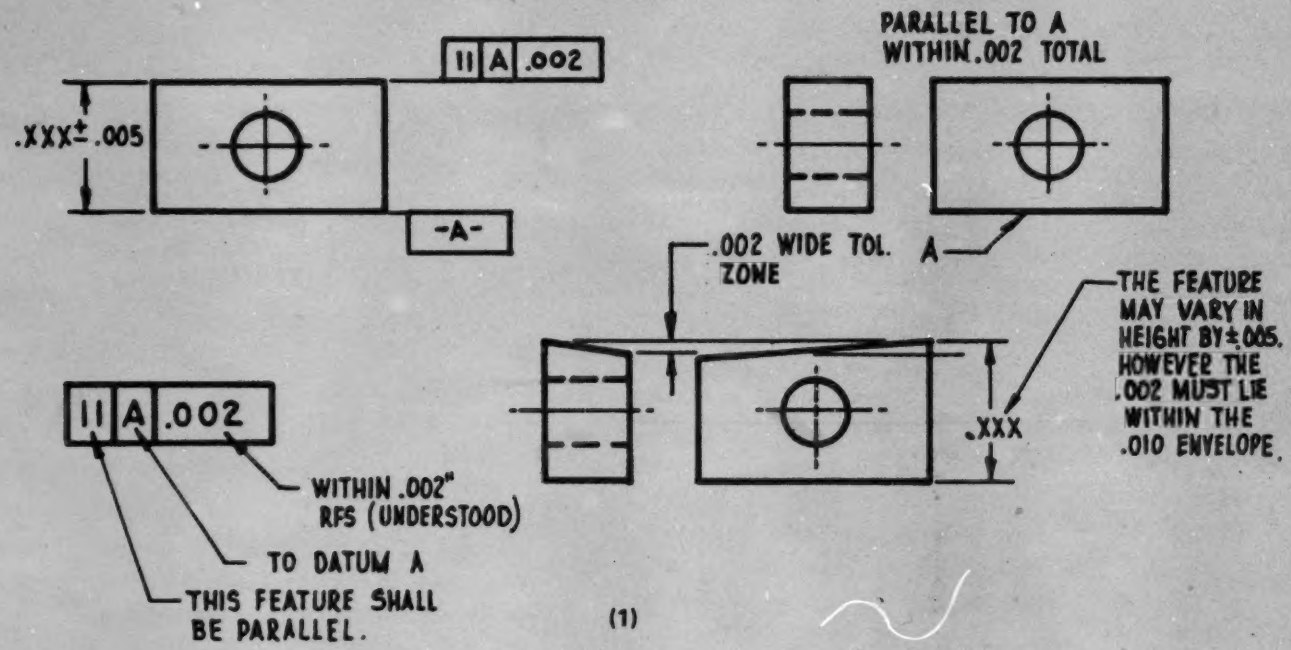
2. Sketch an application of parallelism between two planes, with all tolerances specified.

3. Sketch an application of parallelism when the tolerance zone is a cylinder. Show all applicable tolerances.

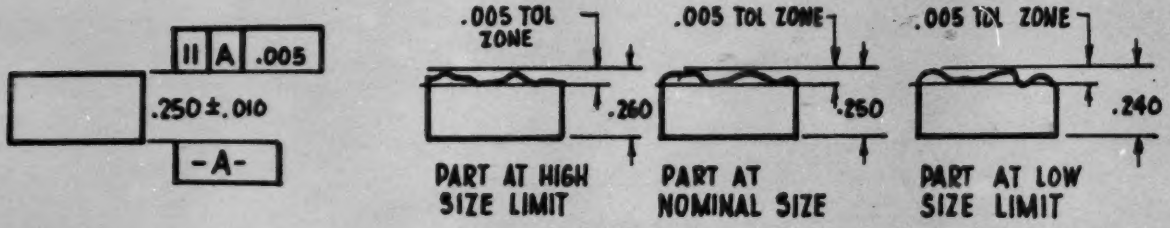
4. Sketch an RFS application and an MMC application. Show all applicable tolerances.

PARALLELISM //

SURFACE TO DATUM FEATURE


// | A | .002
.XXX ± .005
-A-
PARALLEL TO A WITHIN .002 TOTAL
.002 WIDE TOL. ZONE
A
THE FEATURE MAY VARY IN HEIGHT BY ± .005. HOWEVER THE .002 MUST LIE WITHIN THE .010 ENVELOPE.
.XXX
// | A | .002
WITHIN .002" RFS (UNDERSTOOD)
TO DATUM A
THIS FEATURE SHALL BE PARALLEL.

(1)



(2)

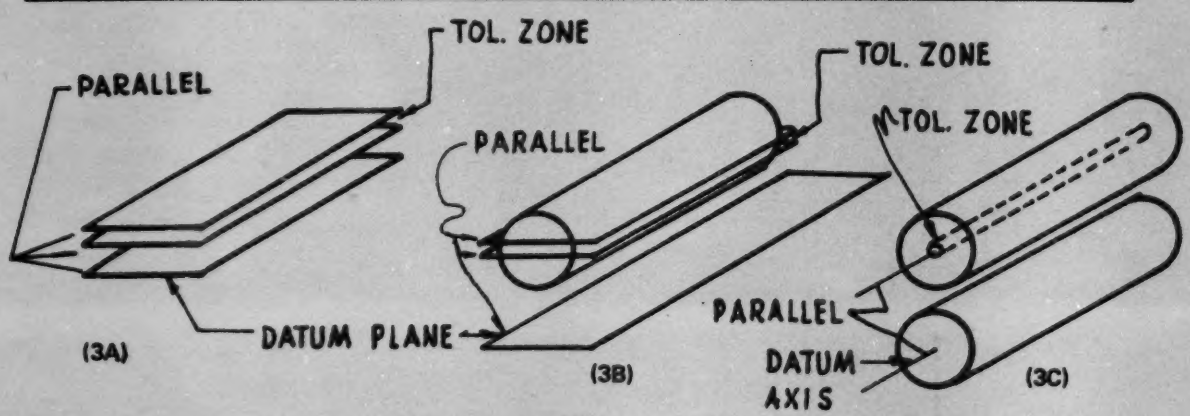


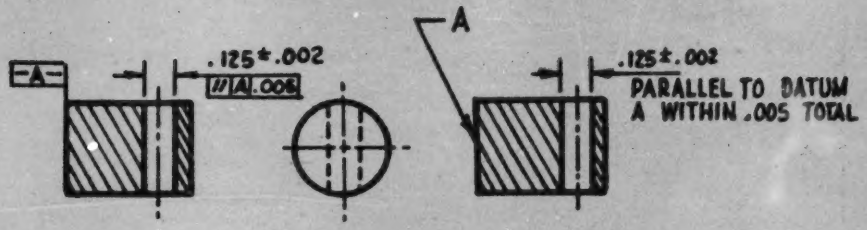
Figure 11

25

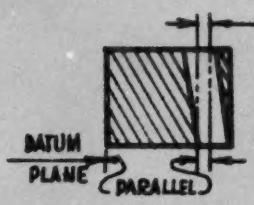
PARALLELISM

FEATURE AT RFS, DATUM A PLANE

EXAMPLE



INTERPRETATION

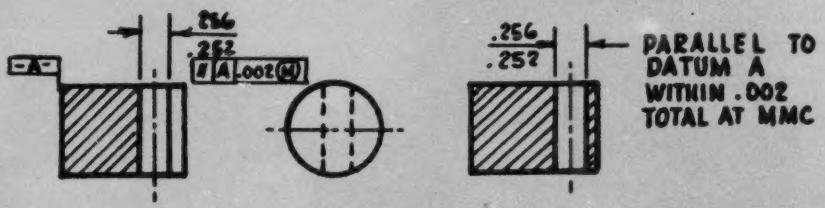


.005 TOL ZONE.
THE FEATURE AXIS MUST BE WITHIN THE SPECIFIED TOLERANCE OF LOCATION REGARDLESS OF THE ACTUAL SIZE OF THE FEATURE, ITS AXIS MUST LIE BETWEEN TWO PARALLEL PLANES, .005 APART, WHICH ARE PARALLEL TO THE DATUM PLANE A.

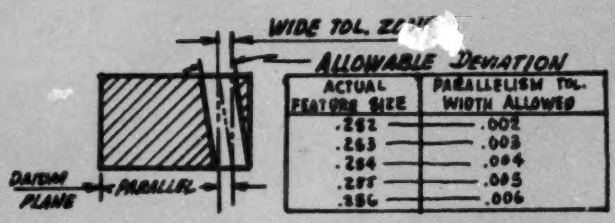
(1)

FEATURE AT MMC, DATUM A PLANE

EXAMPLE



INTERPRETATION



FEATURE AXIS MUST BE WITHIN SPECIFIED TOL. OF LOCATION AND HOLE (AT MMC). AXIS MUST LIE BETWEEN TWO PARALLEL PLANES .002 APART, WHICH ARE PARALLEL TO THE DATUM PLANE A

WHEN HOLE SIZE DEPARTS FROM MMC (GETS LARGER), AN INCREASE IN THE PARALLELISM TOL IS ALLOWED EQUAL TO THE AMOUNT OF DEPARTURE.

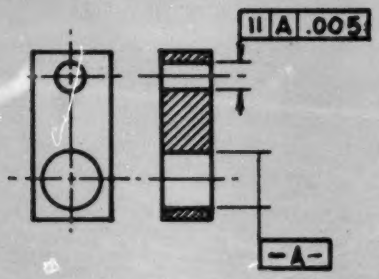
(2)

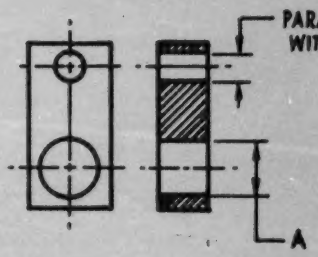
Figure 12

PARALLELISM

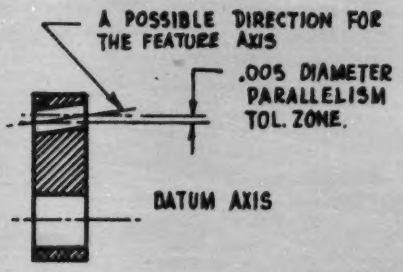
FEATURE AT RFS, DATUM FEATURE AT RFS.

EXAMPLE





INTERPRETATION



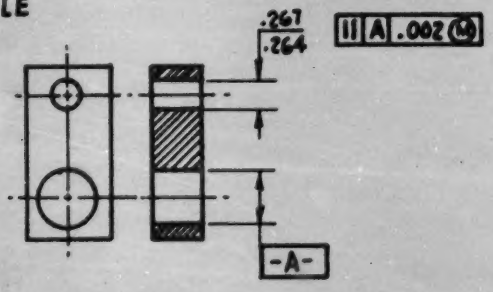
EXPLANATION:

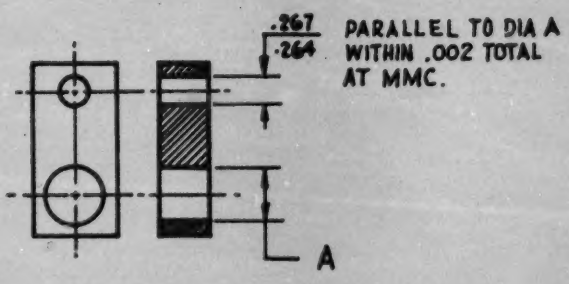
THE FEATURE AXIS MUST BE WITHIN THE SPECIFIED TOLERANCE OF LOCATION. REGARDLESS OF THE HOLE SIZE (THE FEATURE), ITS AXIS MUST LIE WITHIN A CYLINDRICAL ZONE (.005 DIAMETER) WHICH IS PARALLEL TO THE DATUM AXIS.

(1)

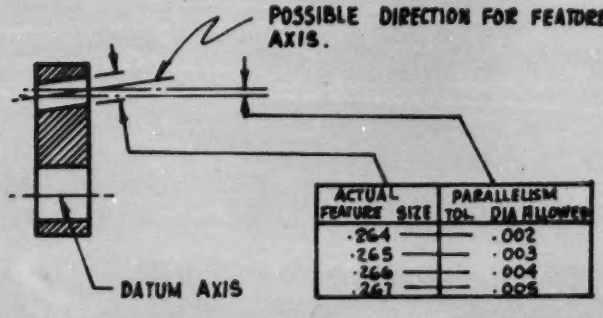
FEATURE AT MMC, DATUM FEATURE AT RFS

EXAMPLE





INTERPRETATION



ACTUAL FEATURE SIZE	PARALLELISM TOL. DIA ALLOWED
.264	.002
.265	.003
.266	.004
.267	.005

WHEN THE FEATURE IS AT MMC (.264), THE MAXIMUM PARALLELISM TOL. IS .002 DIA. WHEN THE FEATURE IS LARGER THAN ITS SPECIFIED MIN. SIZE, AN INCREASE IN THE PARALLELISM TOL. IS PERMISSIBLE.

(2)

Figure 13

PERPENDICULARITY ⊥

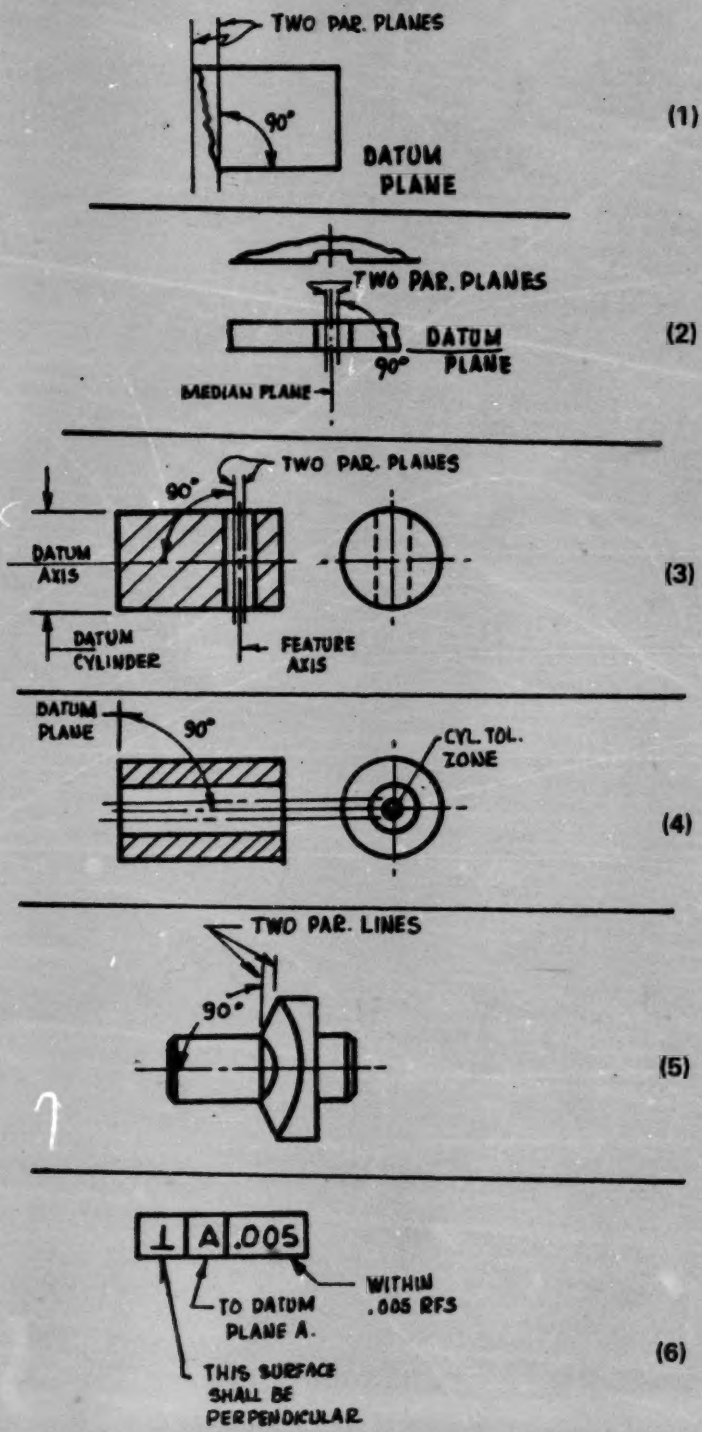


Figure 14

UNIT II
TOLERANCES OF FORM

Lesson 5 — **Perpendicularity**

Objective: You will be able to identify and specify the perpendicularity tolerance.

Related Information

1.0 Perpendicularity Tolerance

1.1 Definition: Perpendicularity is the condition of a surface, axis or line which is exactly 90° from a datum plane, axis, or line.

1.2 A perpendicularity tolerance applied to a surface (with respect to a datum plane) specifies that all elements of the surface must lie between two parallel planes perpendicular to the datum axis. These two parallel planes are separated by a distance equal to the tolerance. Examples of perpendicularity are shown in Figs. 14, 15, and 16.

1.3 A perpendicularity tolerance specifies any of the following:

1. A tolerance zone confined by two parallel planes perpendicular to a datum plane within which

   a. the surface of a feature must lie. See Fig. 14(1).
   b. the median plane of a feature must lie. See Fig. 14(2).

2. A tolerance zone confined by two parallel planes perpendicular to a datum axis, within which the axis of a feature must lie. See Fig. 14(3) for an example.

3. A cylindrical tolerance zone perpendicular to a datum plane within which the axis of the feature must lie. See Fig. 14(4) for an example.

4. A tolerance zone confined by two parallel, straight lines perpendicular to a datum plane or datum axis within which an element of the surface must lie. See Fig. 14(5) for an example.

2.0 Study the examples very carefully and you will notice that almost every aspect of perpendicularity is covered. Applications of the various principles are necessary, because through application you will learn the concepts of this subject. Consider some previous drawings you have made and see if you can apply the principles you have learned to date.

ASSIGNMENT:

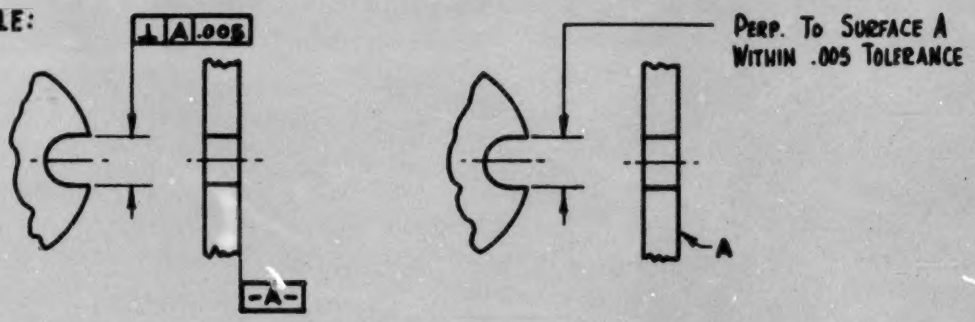
1. Define the term "perpendicularity."

2. Sketch five perpendicularity applications, similar to the five shown in Fig. 14. Show all dimensions and all tolerances.

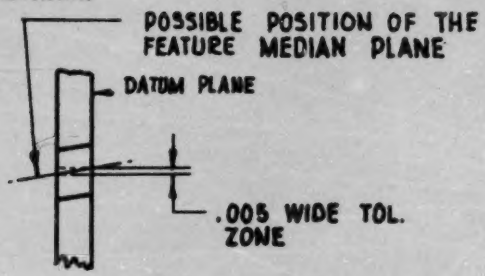
PERPENDICULARITY ⊥

NON- CYLINDRICAL FEATURE AT RFS. DATUM A PLANE

EXAMPLE:



INTERPRETATION

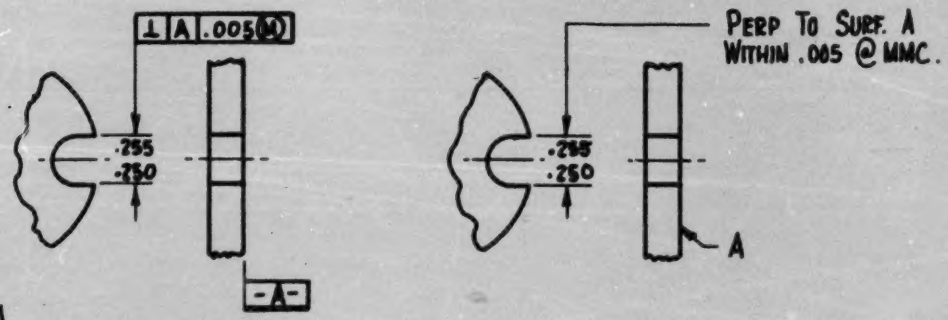


THE FEATURE MEDIAN PLANE MUST BE WITHIN THE SPECIFIED TOL. OF LOCATION. REGARDLESS OF THE ACTUAL SIZE OF THE FEATURE, ITS MEDIAN PLANE MUST LIE BETWEEN TWO PARALLEL PLANES (.005 APART), WHICH ARE PERP. TO THE DATUM PLANE.

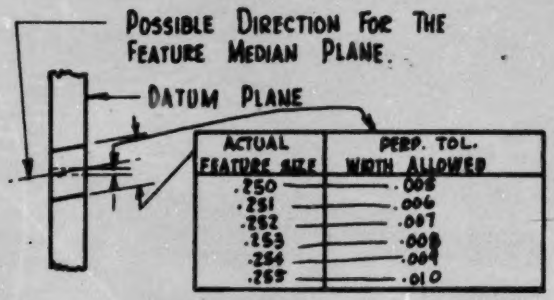
(1)

NON- CYLINDRICAL FEATURE AT MMC, DATUM A PLANE.

EXAMPLE



INTERPRETATION



THE FEATURE MEDIAN PLANE MUST BE WITHIN THE SPECIFIED TOL. OF LOCATION WHEN THE FEATURE IS AT MAXIMUM MAT'L CONDITION (.250) THE MAX PERP. TOL. IS .005 WIDE. WHERE THE FEATURE IS LARGER THAN ITS SPECIFIED MIN. SIZE, AN INCREASE IN PERP. IS ALLOWED. (ADDITIONAL TOL.)

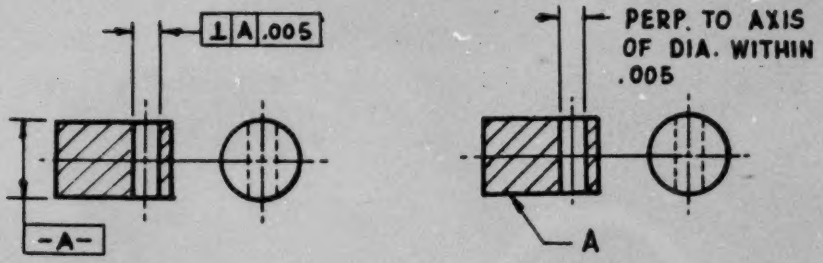
(2)

Figure 15

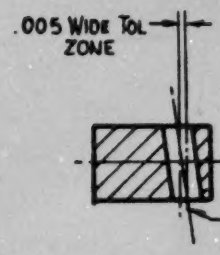
PERPENDICULARITY ⊥

CYLINDRICAL FEATURE AT RFS, DATUM A CYLINDER

EXAMPLE



INTERPRETATION



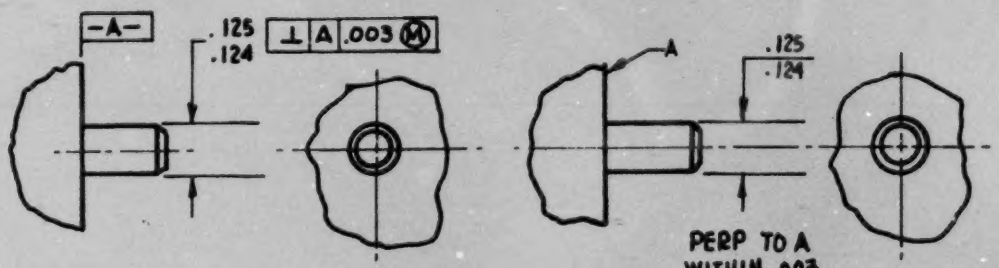
THE FEATURE AXIS MUST BE WITHIN THE SPECIFIED TOL. OF LOCATION & MUST LIE BETWEEN TWO PLANES (.005 APART) WHICH ARE PERP. TO THE DATUM AXIS.

NOTE: THIS TOL. APPLIES ONLY TO THE VIEW ON WHICH IT IS SPECIFIED. IN THE END VIEW, IF IT IS NECESSARY TO CONTROL THE INTERSECTION OF THE FEATURE AXIS & THE DATUM AXIS TO A FINER DEGREE THAN THE SPECIFIED TOL. OF LOCATION, IT SHOULD BE SPECIFIED BY A SEPARATE REQUIREMENT.

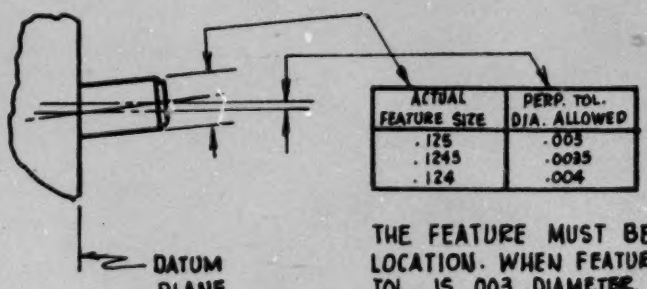
(1)

CYLINDRICAL FEATURE AT MMC, DATUM A PLANE

EXAMPLE



INTERPRETATION.



ACTUAL FEATURE SIZE	PERP. TOL. DIA. ALLOWED
.125	.003
.1245	.0035
.124	.004

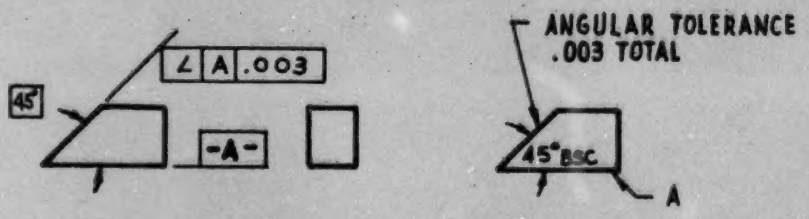
THE FEATURE MUST BE WITHIN THE SPECIFIED TOL. OF LOCATION. WHEN FEATURE IS AT MMC (.125), THE PERP. TOL. IS .003 DIAMETER. AS THE FEATURE SIZE DEPARTS FROM MMC (GETS SMALLER), AN INCREASE IN TOL. IS PERMITTED EQUAL TO THE AMOUNT OF DEPARTURE.

(2)

Figure 16

ANGULARITY ∠
PLANE SURFACE

EXAMPLE



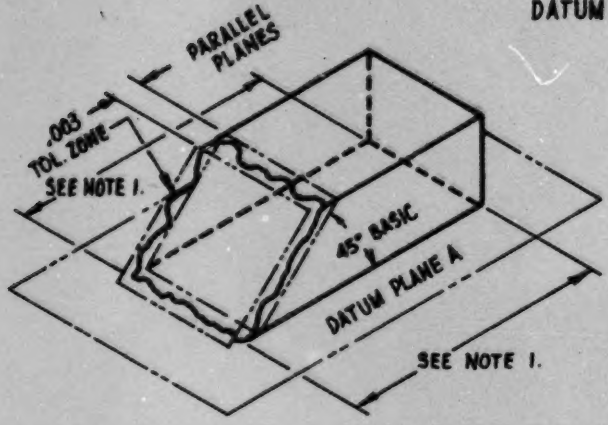
INTERPRETATION

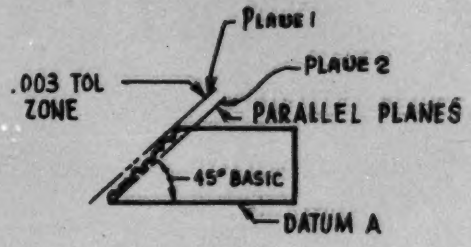
THIS SURFACE SHALL BE AT 45° (THE SPECIFIED ANGLE)

∠ A .003

WITHIN .003 INCH

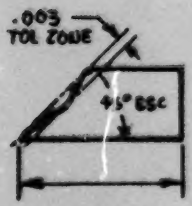
IN RELATION TO DATUM PLANE A

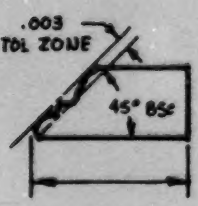




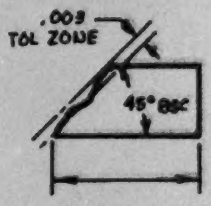
NOTES:

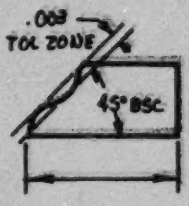
1. THE ANGULAR TOLERANCE ZONE IS COMPOSED OF TWO PARALLEL PLANES .003 APART, @ 45° BASIC TO THE DATUM PLANE A. THE ENTIRE SURFACE MUST FALL WITHIN THE .003 ZONE TO BE ACCEPTABLE. NOTE IN FIGURES BELOW HOW THE TOLERANCE ZONE IS ESTABLISHED.





PART AT HIGH SIZE LIMIT OF TOLERANCE





PART AT LOW SIZE LIMIT OF TOLERANCE

Figure 17

UNIT II
TOLERANCES OF FORM

Lesson 6 **Angularity**

Objective: You will be able to identify and specify the angularity tolerance.

Related Information:

1.0 Angularity Tolerance – Plane Surfaces

1.1 Definition: Angularity is the condition of a surface, axis, or center plane which is at the specified angle (other than 90°) from a datum plane or axis.

1.2 Angularity tolerance is the distance between two parallel planes, inclined at the angle specified to a datum plane or axis, within which the toleranced surface, axis, or center plane must lie.

1.3 The example in Fig. 17 shows a part with an angular tolerance. The symbol is interpreted as "this surface shall be at an angle of 45° with respect to datum A within .003."

1.4 To be acceptable, the entire surface must fall between two parallel planes, (1) and (2), which are .003 apart. At the bottom of Fig. 17 you will notice how the tolerance zone is established.

2.0 Angularity Tolerance – Center Plane

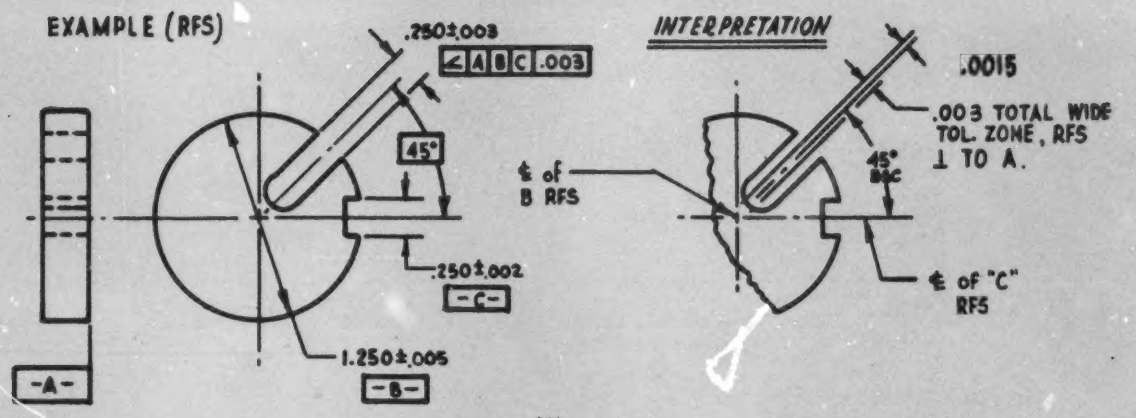
2.1 Where angularity of a feature with *size* is to be specified, the center (median) plane or axis of the feature becomes the basis for control. In this case, a *form* tolerance characteristic is used as a positional tolerance control.

2.2 In the example in Fig. 18(1), the slot is an RFS application. There exists an angular relationship to three datums. The produced slot median or center plane must be within the .003 total wide tolerance zone regardless of the actual size of the slot or datum. The tolerance zone extremities are defined as two parallel planes parallel to the median plane and perpendicular to datum A, with the center of the median plane angle established by the centers of datums B and C, RFS.

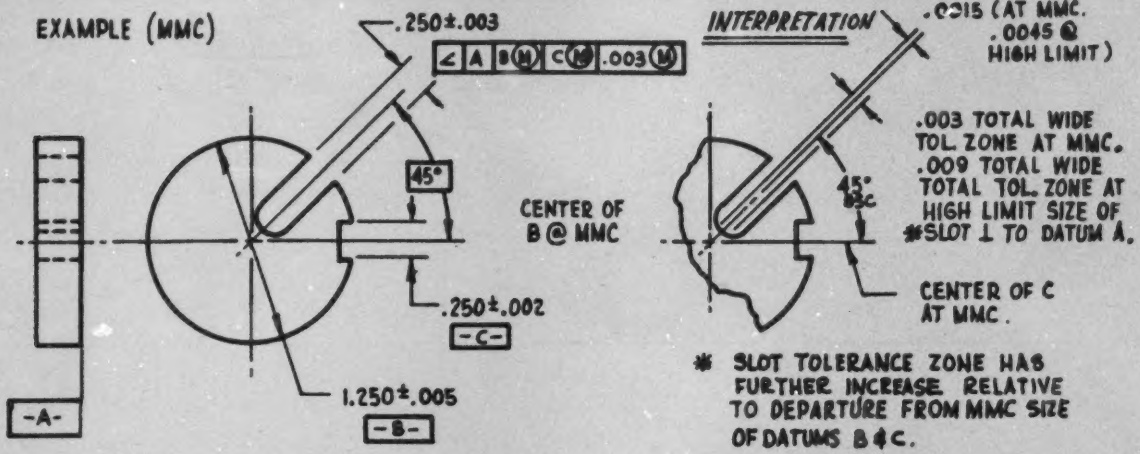
2.3 The example in Fig. 18(2) illustrates an MMC application. Again we have a slot in an angular relationship to three datums. The produced slot median or center plane must lie within the .003 total wide tolerance zone when slot and datums B and C are at MMC sizes (slot .247, B at 1.255, and outer slot C at .248). The tolerance zone extremities are defined as two parallel planes parallel to the

ANGULARITY ∠

CENTER PLANE

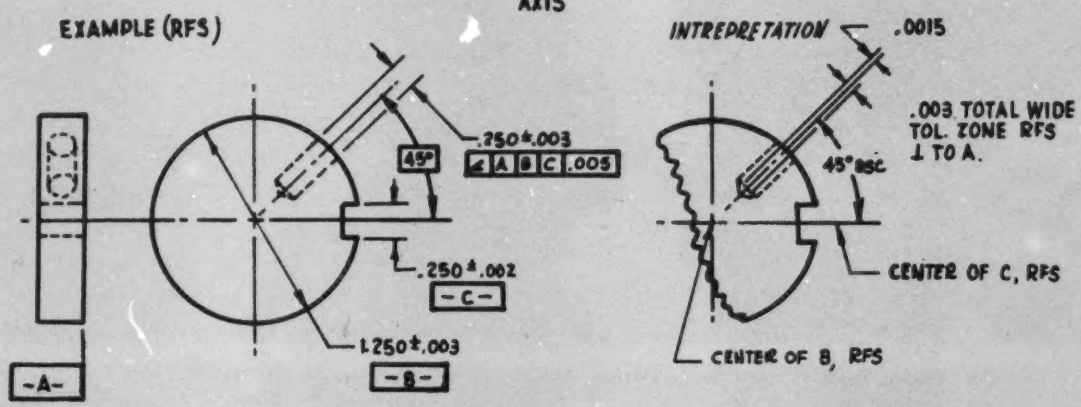


(1)


EXAMPLE (MMC)
.250±.003
∠ A B(M) C(M) .003(M)
45°
.250±.002
-C-
1.250±.005
-B-
-A-
CENTER OF B @ MMC
INTERPRETATION
.0015 (AT MMC. .0045 @ HIGH LIMIT)
.003 TOTAL WIDE TOL. ZONE AT MMC. .009 TOTAL WIDE TOTAL TOL. ZONE AT HIGH LIMIT SIZE OF *SLOT ⊥ TO DATUM A.
45° BSC
CENTER OF C AT MMC.
* SLOT TOLERANCE ZONE HAS FURTHER INCREASE RELATIVE TO DEPARTURE FROM MMC SIZE OF DATUMS B & C.

(2)

AXIS



(3)

Figure 18

median plane angle established by the centers of datums B and C at MMC size. As the inside slot departs from MMC size (gets larger), an amount equal to that departure is added to the total wide tolerance zone of the slot. At the slot high limit, the total wide tolerance zone is .009 (± .003 = .006 + .003M = .009). As the datums B and C depart from MMC size, further tolerance is added to the total tolerance zone of the slot. From the geometry of this part, the amount of added tolerance available relative to the datum departures from MMC size is difficult to calculate.

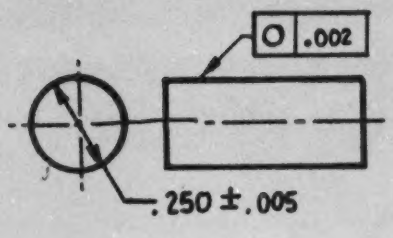
2.4 No. 3 on Fig. 18 illustrates an RFS application of a hole in an angular relationship with three datums. The produced hole axis must lie within the .005 total wide tolerance zone regardless of the actual size of the hole or the datums. The tolerance zone extremities are defined as two parallel planes parallel to the axis and perpendicular to datum A, with the vertex of the axis angle established by the centers of datums B and C, RFS.

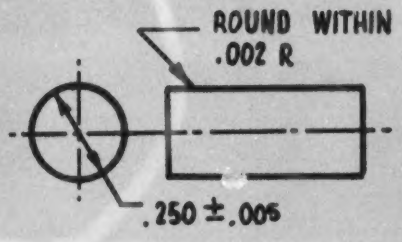
ASSIGNMENT:

1. Define the angularity tolerance.
2. Sketch an example of angularity, plane surface only. Show all datums, tolerances, and dimensions.
3. Sketch an example of angularity, using MMC. Show all tolerances and dimensions.
4. Sketch an example of angularity, using RFS. Show all tolerances and dimensions.

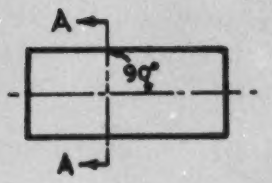
ROUNDNESS OF A CYLINDER ○

EXAMPLE





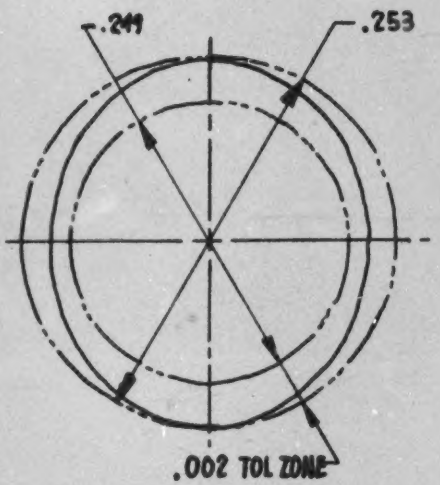
INTERPRETATION



THIS FEATURE SHALL BE ROUND

WITHIN .002 WIDE TOL. ZONE.

SECTION A-A



THE OUTSIDE SURFACE AT ANY CROSS SECTION PERPENDICULAR TO THE AXIS MUST BE WITHIN THE SPECIFIED TOLERANCE OF SIZE & MUST LIE BETWEEN TWO CONCENTRIC CIRCLES, ONE HAVING A RADIUS .002 LARGER THAN THE OTHER.

Figure 19

UNIT II
TOLERANCES OF FORM

Lesson 7 — **Roundness (Circularity)**

Objective: You will be able to identify and specify the roundness tolerance.

Related Information:

1.0 Roundness Tolerance

1.1 Definition: Roundness is the condition on a surface of revolution (cylinder, cone, sphere) where all points of the surface intersected by any plane (1) perpendicular to a common axis (cylinder, cone) or (2) passing through a common center (sphere) are equidistant from the axis or center.

1.2 Paragraph 1.1 is not as difficult to understand as you may think. Consider a cylinder (for example, a long piece of pipe), and imagine the outermost surface of the cylinder as being bound by a perfect circle, and the innermost surface being bound by a perfect circle. These two circles form a tolerance band within which the part in question must fall.

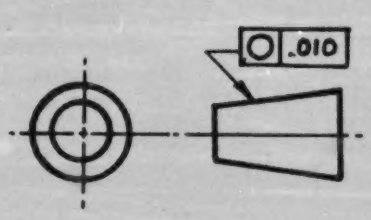
1.3 The illustration at the top of Fig. 19 shows a cylindrical part with a roundness of .002 specified. The interpretation shows how you establish the .002 tolerance zone. Note that the tolerance zone is the width of the ring between the two concentric circles.

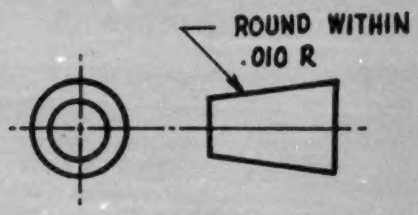
A roundness tolerance zone is established relative to the actual size of the part when measured at the outside surface at any cross section perpendicular to the part axis. Roundness tolerance applies only at the cross-sectional point of measurement, and is only relative to the size at that point. Therefore, a cylindrical part with roundness tolerance control could taper and still meet the roundness requirement.

1.4 Referring again to Fig. 19, it has been assumed that the part measures a maximum of .253 at a particular cross section. The .002 roundness tolerance zone then is established by two perfect concentric circles, one at the .253 diameter and the other (.004 smaller) at the .249 diameter. This establishes the tolerance zone of *.002 width* between the concentric circles. To be acceptable, the part surface at that cross section must fall within the .002 wide tolerance zone.

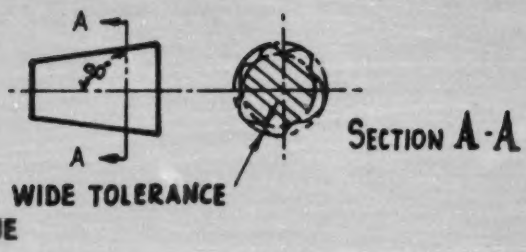
ROUNDNESS OF A CONE ○

EXAMPLE





INTERPRETATION

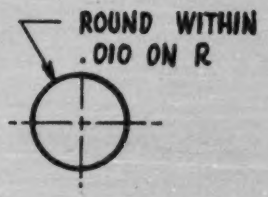


THE OUTSIDE SURFACE AT ANY CROSS SECTION PERPENDICULAR TO THE AXIS MUST BE WITHIN THE SPECIFIED TOLERANCE OF SIZE AND MUST LIE BETWEEN TWO CONCENTRIC CIRCLES (ONE HAVING A RAD. .010 LARGER THAN THE OTHER).

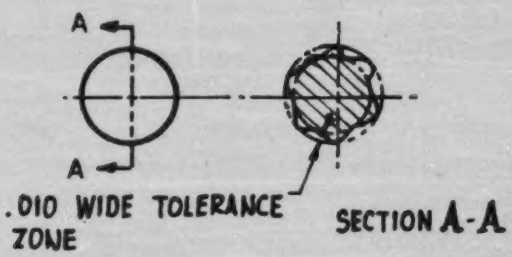
(1)

ROUNDNESS OF A SPHERE

EXAMPLE



INTERPRETATION



THE OUTSIDE SURFACE AT ANY CROSS SECTION PASSING THROUGH A COMMON CENTER MUST BE WITHIN THE SPECIFIED TOL. OF SIZE AND MUST BE BETWEEN TWO CONCENTRIC CIRCLES (ONE HAVING A RADIUS .010 LARGER THAN THE OTHER). THEREFORE THE SURFACE MUST LIE BETWEEN TWO CONCENTIC SPHERES SEPARATED .010 APART.

(2)

Figure 20

1.5 A roundness tolerance cannot be modified by an MMC application, since it controls surface elements only. Also a roundness tolerance is a perfect counterpart of itself; therefore no datum references are required or proper. You may compare a roundness tolerance to a straight tolerance zone circled around a circle.

2.0 Roundness of a Cone

2.1 No. (1) in Fig. 20 shows a cone-shaped part for which a roundness tolerance of .010 is specified. The outer surface at any cross section perpendicular to the axis must be within the specified tolerance of size and must lie between two concentric circles, one having a radius .010 larger that the other.

3.0 Roundness of a Sphere

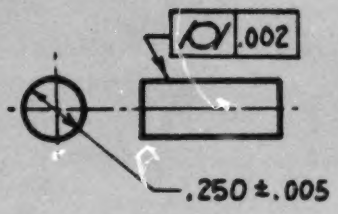
3.1 No. (2) in Fig. 20 illustrates a spherical part. The same interpretation as above applies, except that the tolerance-control reference is to "any cross section passing through the center," rather than "any cross section perpendicular to the axis."

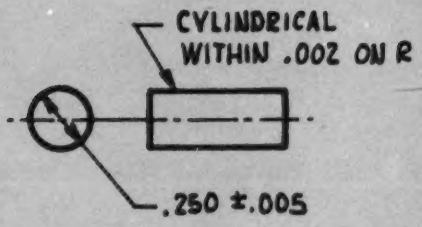
ASSIGNMENT:

1. Define "roundness tolerance," and interpret this definition in your own words.
2. Sketch a roundness tolerance application which is perpendicular to a common axis. Show all dimensions and tolerances.
3. Same as above except passing through a common center.
4. Can roundness tolerance be modified by MMC and RFS?
5. Is it proper to reference roundness to a datum surface or line?

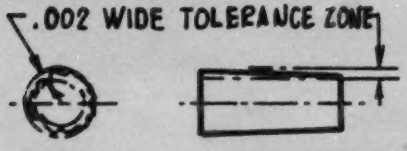
CYLINDRICITY ⌭

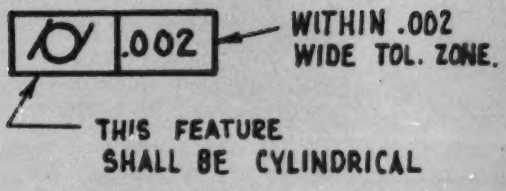
EXAMPLE





INTERPRETATION





THE FEATURE MUST BE WITHIN THE SPECIFIED TOL. OF SIZE AND MUST LIE BETWEEN TWO CONCENTRIC CYLINDERS (ONE HAVING A RAD. .002 LARGER THAN THE OTHER).

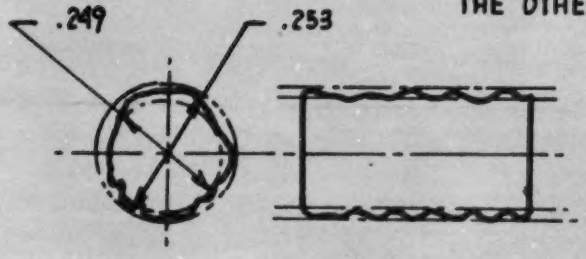


Figure 21

UNIT II
TOLERANCES OF FORM

Lesson 8 — **Cylindricity**

Objextive: You will be able to identify and specify the cylindricity tolerance.

Related Information:

1.0 Cylindricity

1.1 Definition: Cylindricity is the condition of a surface of revolution of one line about another to which it is parallel.

1.2 From Lesson 7 you will recall that *roundness* applies only to that particular cross section where you are considering the roundness. With cylindricity, you are bound by the same two concentric circles; however, now the *whole length* of the part (the full cylinder) is included.

1.3 Cylindricity tolerancing can be applied only to cylindrical forms.

1.4 Cylindricity tolerance simultaneously controls roundness, straightness, and parallelism of the elements of the cylindrical surface. Note how the symbol includes all these concepts.

1.5 Examine the example in Fig. 21. As was previously stated, the cylindricity tolerance applies to the *entire* cylindrical surface as opposed to the cross-sectional measurement considered in roundness. The part, you will notice, has a diameter of .250 ± .005, with a cylindricity tolerance of .002. This .002 tolerance is the "empty" cylinder that goes all around, which has a wall thickness of .002. In the example, let us assume that the largest diameter is .253. This .253 is the diameter of the largest of the concentric cylinders defining the tolerance zone. The smaller of the concentric cylinders is .253 minus the amount of the cylindricity tolerance (.002 on R = .004 on dia.), i.e., .249 dia. The cylindricity tolerance zone is, therefore, *.002 wide* between the concentric cylinders. The entire surface part must fall within this tolerance zone to be acceptable.

1.6 A cylindricity tolerance is a form control of a surface element, and no datum reference is required or proper.

ASSIGNMENT:

1. Define the cylindricity tolerance, and interpret this definition in your own words.

2. Is a datum reference required for cylindricity tolerance?

3. What is the symbol for cylindricity?

4. How does cylindricity differ from roundness tolerance?

5. Sketch a cylindricity-tolerance application with all applicable dimensions and tolerances.

UNIT II
TOLERANCES OF FORM

Lesson 9 — **Profile Tolerancing**

Objective: You will be able to identify and specify profile tolerances.

Related Information:

1.0 Profile Tolerancing

1.1 Definition: Profile tolerancing is a method used to control a uniform amount of variation of a surface or line element of a surface.

1.2 A profile tolerance (either bilateral or unilateral (see Fig. 22) specifies a tolerance zone always measured normal (perpendicular) to the basic profile (applied to the view in which drawn) at all points of the profile, within which the specified part surface profile or line profile must be.

1.3 Profile tolerancing is an excellent method of controlling an irregular curve, arc, or other unusual-part profile. These tolerances are usually applied to surface features, but may also be applied to a line (as part of a feature surface). Profile tolerances must be shown on the appropriate view of a drawing, as follows:

a. An appropriate view or section is drawn which includes the desired profile.

b. The profile is dimensioned by basic dimensions. These dimensions may be location radii, angles, or rectangular coordinate dimensions to a point on the profile.

c. At a point which stands out along the profile, a tolerance zone is shown and dimensioned by one or two phantom lines drawn parallel to the profile. Exaggerate the distance between the phantom lines for clarity. The zone may be shown divided bilaterally or unilaterally. (See Fig. 22(1).)

d. Other necessary dimensioning, as well as the applicable feature-control symbol or note, is added. The symbol or note should be applied in a view in which the surface or line to be controlled is shown as a profile, and which in contour represents the desired feature orientation and relationship.

e. When some limits are expressed on a drawing by a profile tolerance and others by regular tolerance dimensions, the extent or limitation of the profile tolerance must be clearly indicated. Do this by reference letters applied at the extremities of the profile controlled and a notation (e.g., between A and B) added with the profile tolerance symbol or notes.

1.4 Profile tolerancing is a control of *surface form*, and no modifier (MMC) can be used on the feature controlled. Where a datum reference is used, it is implied as RFS.

2.0 Profile of a Surface

2.1 The profile of a surface controls the entire surface, or all the elements of the surface, within a uniform tolerance zone as established from the basic profile. See Fig. 23.

3.0 Profile of a Line

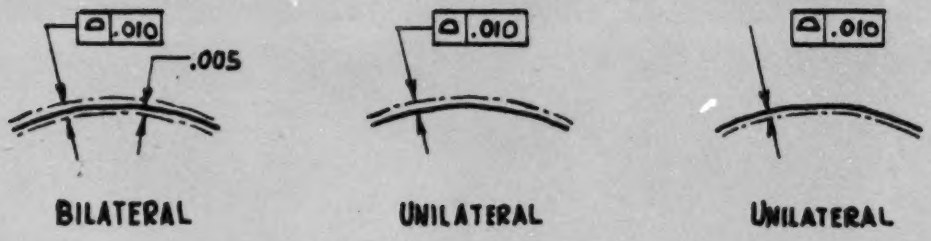
3.1 Line profile control is applied in a manner similar to the application of surface profile. Basic dimensions are used to define the perfect profile as a reference for the tolerance zone established. The basic difference between surface and line profile controls is that line control must be used as a refinement of other controls, e.g., size or surface profile. See Fig. 24 for profile of a line.

4.0 Profile tolerances are best understood by studying the various examples shown in Figs. 22, 23, and 24. Study the examples carefully and note the interpretations.

ASSIGNMENT:

1. Define the profile tolerance.
2. What is a bilateral tolerance?
3. What is a unilateral tolerance?
4. Can profile tolerancing be modified by MMC?
5. Sketch the profile of a line application, showing all dimensions and tolerances.
6. Sketch the profile of a surface application, showing all dimensions and tolerances.

PROFILE TOLERANCE

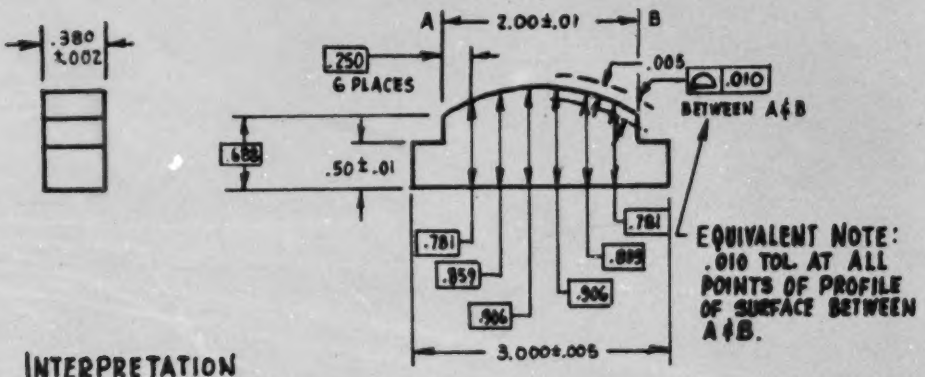


BILATERAL UNILATERAL UNILATERAL

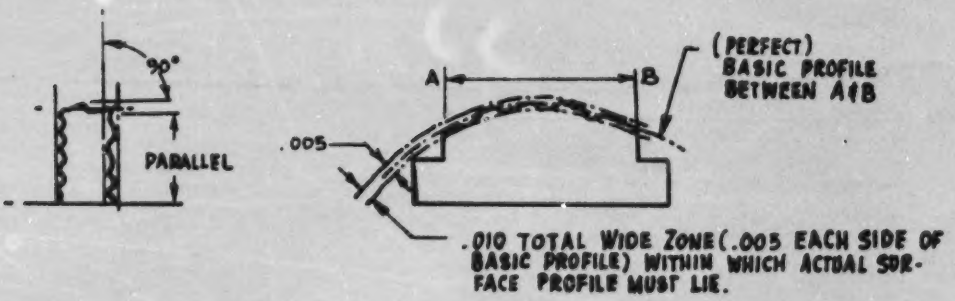
(1)

PROFILE OF A SURFACE ⌓

EXAMPLE



INTERPRETATION


90°
PARALLEL
A
B
(PERFECT) BASIC PROFILE BETWEEN A & B
.005
.010 TOTAL WIDE ZONE (.005 EACH SIDE OF BASIC PROFILE) WITHIN WHICH ACTUAL SURFACE PROFILE MUST LIE.

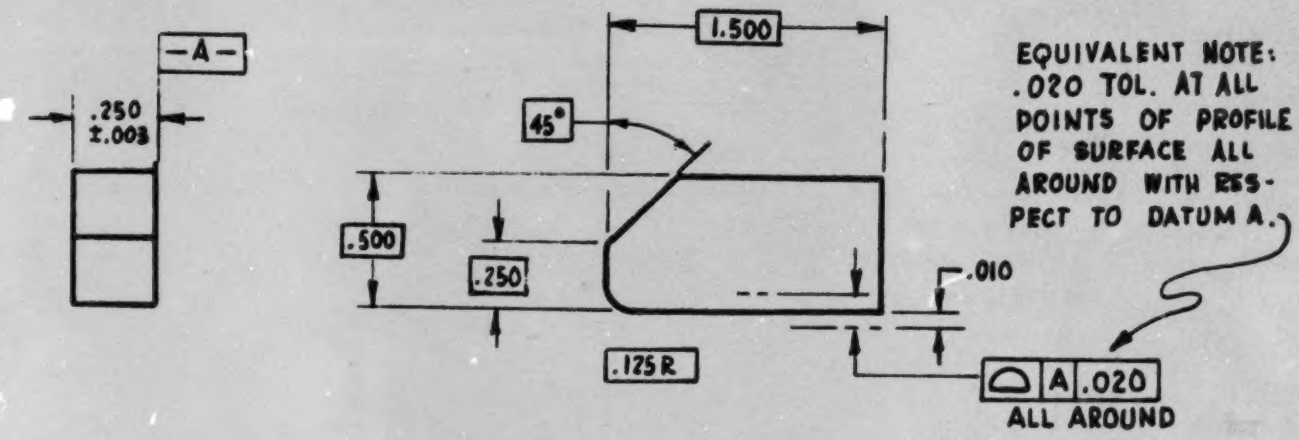
(2)

Figure 22

PROFILE TOLERANCE

PROFILE OF A SURFACE

EXAMPLE



INTERPRETATION

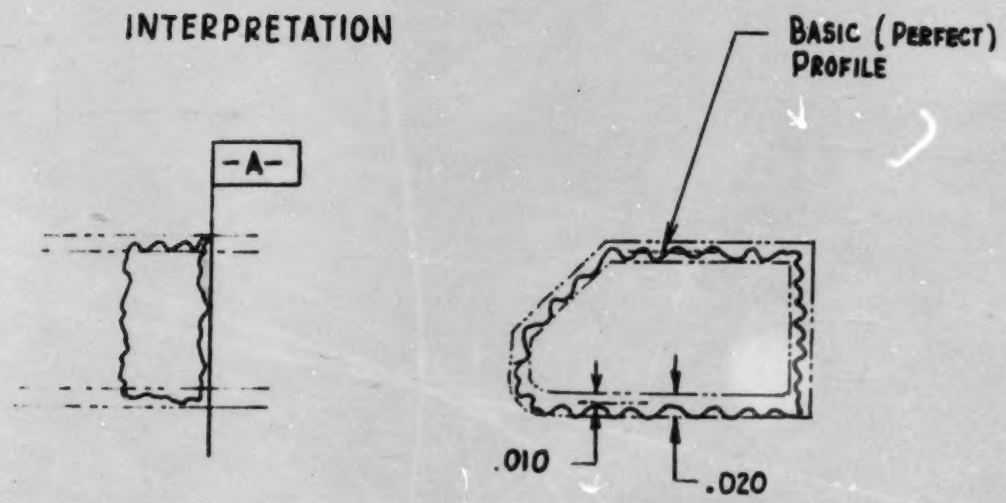
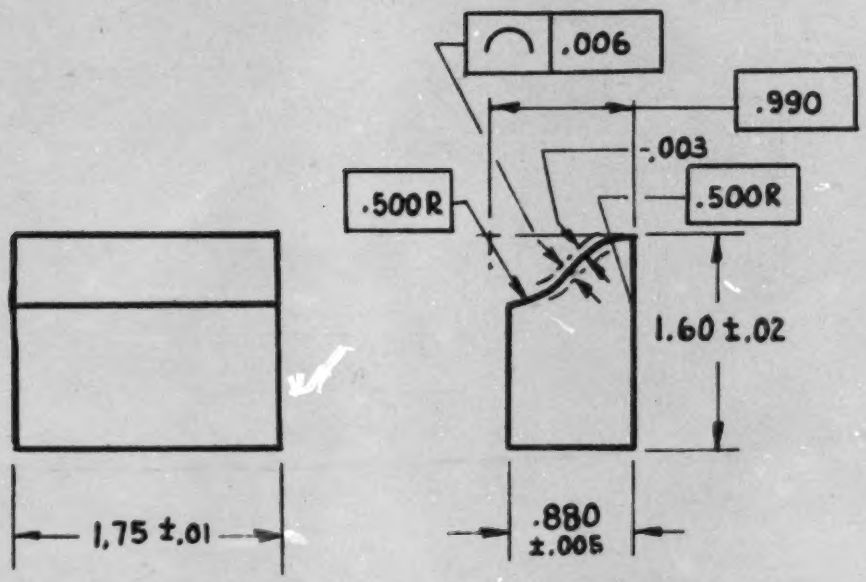


Figure 23

PROFILE TOLERANCE

PROFILE OF LINE ⌒

EXAMPLE



INTERPRETATION

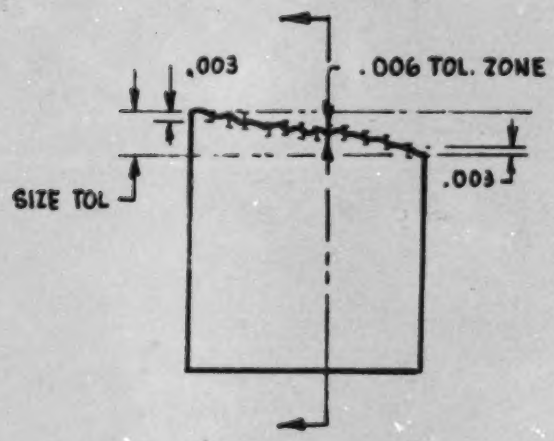
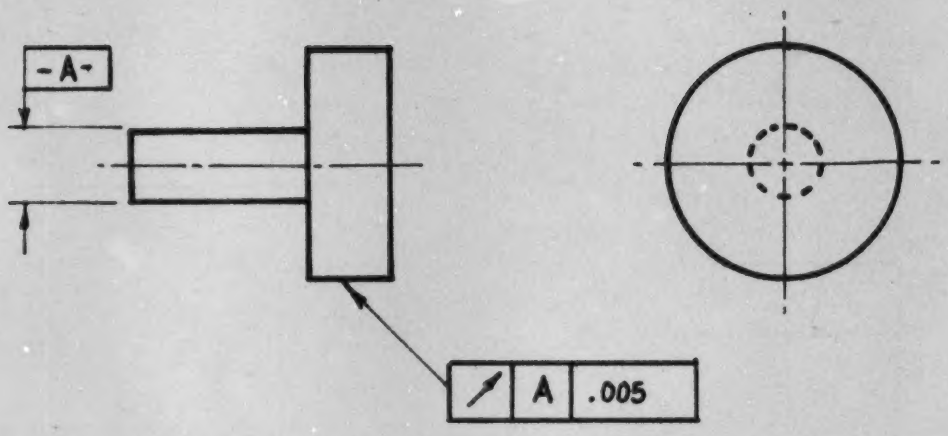


Figure 24

RUNOUT TOLERANCE ↗

EXAMPLE



INTERPRETATION

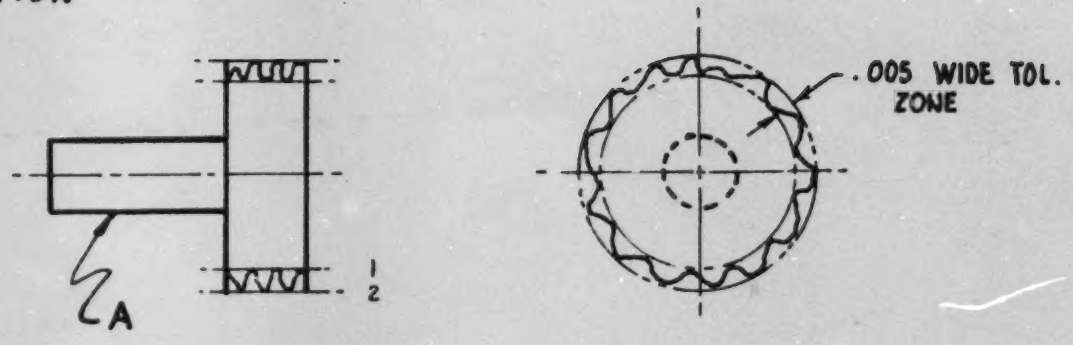


Figure 25

Lesson 10 **What Is Runout?**

Objective: You will be able to identify and specify the runout tolerance.

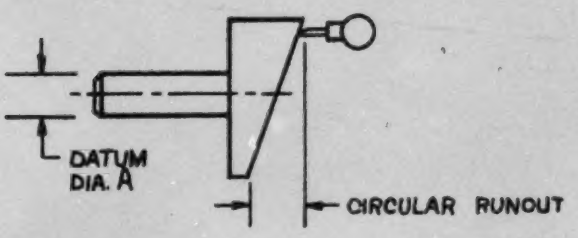
Related Information:

1.0 **Runout**

1.1 Definition: Runout is the deviation from perfect form of a part surface of revolution detected by a full rotation (a complete turn of 360°) of the part on a datum axis, using a dial indicator or equivalent measuring device.

1.2 A runout tolerance is always applied to a cylindrical surface with respect to a datum axis. This means that the entire cylindrical surface must lie between two cylinders which are *coaxial* with the specified datum axis. These two cylinders are separated by a distance equal to the tolerance zone. Runout, as was stated in the definition, is detected by rotating the part about the datum axis and recording the excursions on a dial indicator applied to the surface. Runout differs from cylindricity: in cylindricity a datum axis was not proper; therefore it was not specified.

1.3 In the example in Fig. 25, you will find the basis of control of runout tolerances. Notice the datum axis (the part center line), and that the features which we are trying to control are all measured from this datum axis. Surfaces at right angles to the datum axis can also be controlled with the runout tolerances. (See Fig. 26.)



Where the surface is flat but not perpendicular to dia A, circular runout exists.

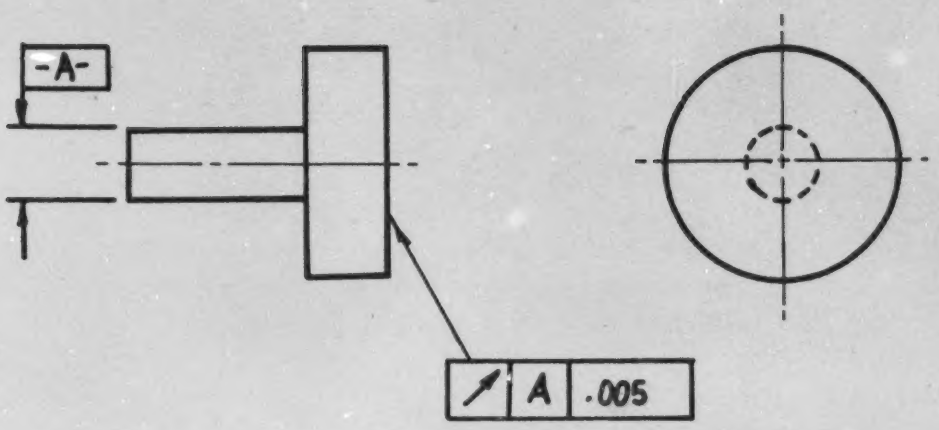
2.0 The concept of runout is very difficult to explain and also very difficult for the reader to comprehend. Further explanations and illustrations are therefore omitted, because it is the writer's contention that, if the basics are well understood, the other more difficult applications will be understood as the draftsman gets more experience.

ASSIGNMENT:

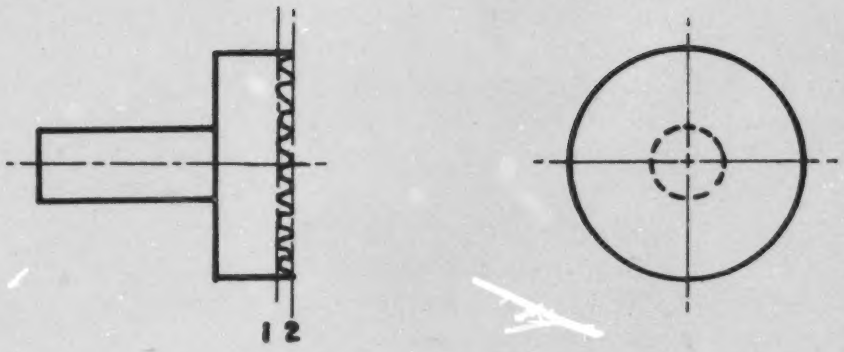
1. Define the runout tolerance.
2. Sketch an application of runout, showing all datums and dimensions.
3. Is a datum necessary when using runout as a form control?
4. What is the runout symbol?
5. Define runout in your words.

RUNOUT TOLERANCE

EXAMPLE



INTERPRETATION



PLANES 1 AND 2 ARE PERPENDICULAR TO DATUM A. WITHIN .005.

Figure 26

UNIT III
TOLERANCES OF POSITION

Lesson 1 — **Defining True Position**

Objectives: You will be able to

1. Identify the three tolerances of position.
2. Explain the meaning of true position and true-position tolerancing.

Related Information:

1.0 True Position

1.1 Definition: True position is a term used to describe the perfect (exact) location of a point, line, or plane (normally the center of a feature) as related to a datum reference or other feature.

1.2 Tolerances of position include three concepts: true position, concentricity, and symmetry.

1.3 A true-position tolerance is the total permissible variation in the location of a feature about its true position. For cylindrical features (holes and bosses), the true-position tolerance is the diameter (cylinder) of the tolerance zone around the true position. For other features (slots, tabs, etc.) the true-position tolerance is the *total width* of the tolerance zone within which the center plane of the feature must lie, the center plane of the zone being at the true position.

1.4 In the true-position dimensioning system, the draftsman selects those features (datum features) that are most important for the functioning of the part as the reference for the location of the other elements of the part.

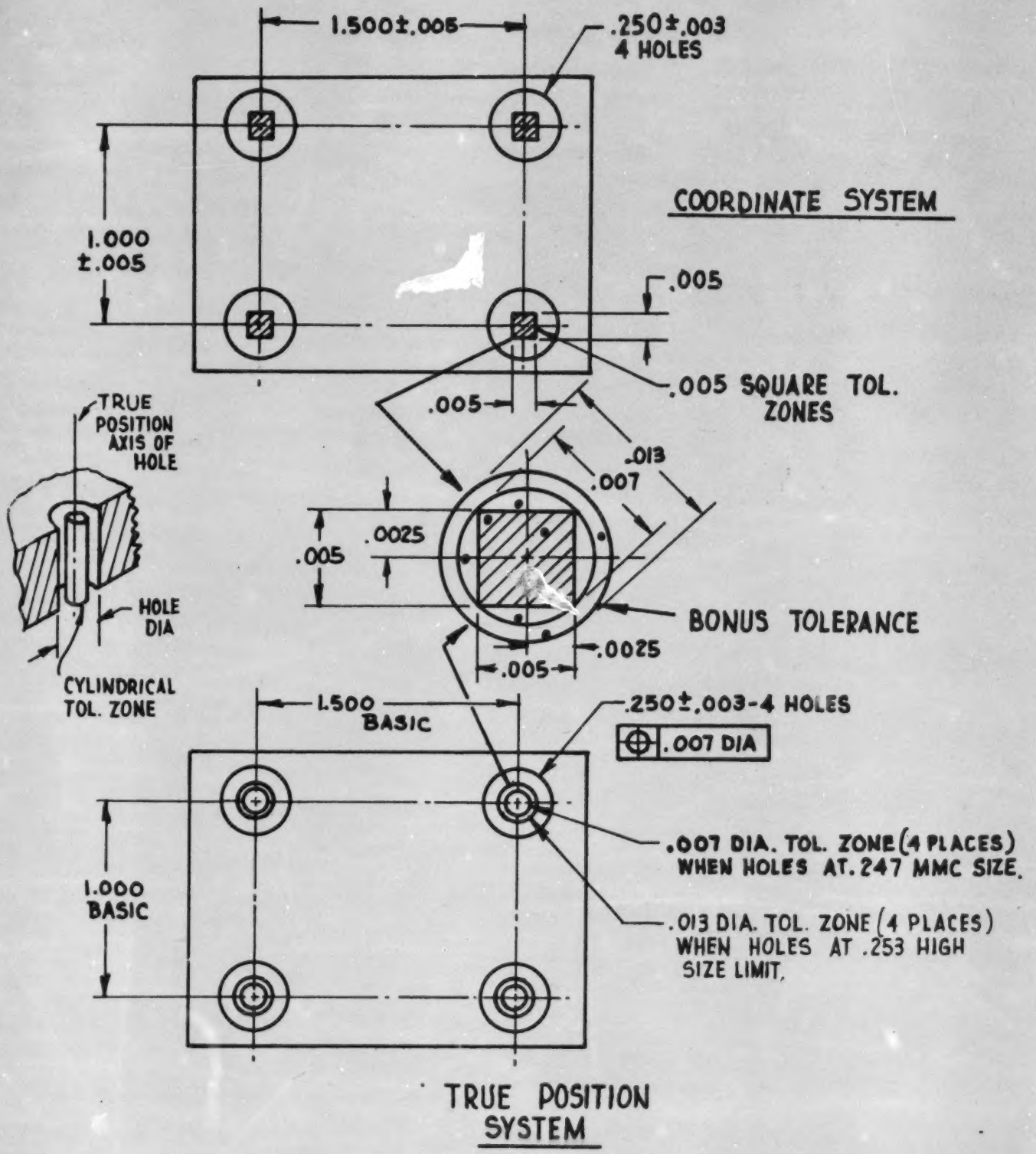
1.5 Notice in the above paragraph that the part *function* was mentioned. True-position datums are selected with the part function in mind. Enough datum features are chosen to position the part in relation to a set of three mutually perpendicular planes (datum planes).

1.6 In the true-positioning system, the location of the center line of holes is found by specifying the true dimension to the center from the datum features, and then specifying the distance that the center may vary from this true position.

ASSIGNMENT:

1. Give a definition for true position in your own words.
2. What are the three tolerances of position?
3. What are features, and how does the draftsman use features in the true-position system?
4. What is the tolerance zone for holes?
5. What is the tolerance zone for slots and tabs?

TRUE POSITION THEORY



TRUE POSITION SYSTEM

Figure 27

UNIT III
TOLERANCES OF POSITION

Lesson 2 — **True Position Theory**

Objective: You will be able to identify the concepts of true-position theory.

Related Information:

1.0 In order to understand the theory of true position, a comparison will be made with the old system (rectangular-coordinate system).

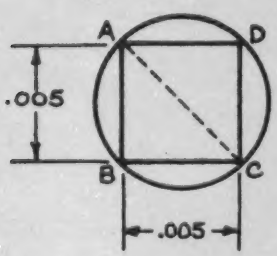
1.1 Imagine a part with four holes in a pattern which must line up with a mating part to accept screws.

The top illustration in Fig. 27 shows the part with a hole-pattern dimensioned and toleranced using the coordinate system. The bottom illustration in the figure shows the same part dimensioned using the true-position system. Comparing the two approaches, we find the following differences:

1. The top figure shows four *squares.* These are the result of tolerance going first plus .005 then minus .005 in the horizontal plane, and then plus or minus .005 in the vertical plane. However, the tolerance zone in true position is *round.*

2. The hole-center location tolerance in the top figure is a part of the coordinates (the 1.500 and the 1.000 dimensions). In the bottom figure, however, the location tolerance is associated with the hole-size dimension and is shown in the feature-control symbol at the right. The 1.500 and the 1.000 coordinates are retained in the true-position application, but are now stated as *basic* or exact values.

1.2 For true-position tolerancing, the .005 square tolerance zone has to be converted to an equivalent true-position tolerance zone. This is accomplished in the following manner:



Diagonal $AC = \sqrt{AB^2 + BC^2}$

$AC = \sqrt{(.005)^2 + (.005)^2}$

$AC = \sqrt{50 \times 10^{-6}}$

$AC = .007$ inches

The two tolerance zones are superimposed on each other in the enlarged detail in the center of the figure.

1.3 The black dots represent eight possible inspected centers of this hole. If the coordinate-zone tolerance is applied, only three dots fall into the square zone; thus three parts out of eight pass inspection.

1.4 The true-position axis is a cylinder, however, and a cylindrical hole should normally have a cylindrical tolerance zone. Look again at the black dots and their position with respect to the desired exact location. The drawing clearly shows the inconsistencies of the coordinate system. Five dots fall outside the square zone, but examination of the drawing will show how inconsistent it is to reject all of them.

1.5 The dot in the upper-left diagonal corner of the square zone, and the dot on the left – outside the square zone – are in reality at nearly the same distance from the desired exact center. However, in terms of the square coordinate zone, the hole on the left is unacceptable by a wide margin, whereas the upper-left hole is acceptable.

1.6 Note that a hole produced off center under the coordinate system has a greater tolerance if the shift is on the diagonal rather than in the horizontal or vertical direction.

1.7 The normal function of a hole relates to its mating feature in any direction. From the example just explained, we can see that the square zone restriction seems unreasonable and incorrect. Thus the true-position tolerance zone, which recognizes and accounts for location of round or cylindrical features as they relate to one another, is more realistic and practical.

1.8 True position in normal applications is derived from design requirements and not from converted coordinates. This is a bonus you really get with this system. The maximum material sizes of the features (hole and mating component) are used to determine this tolerance.

1.9 This tolerance, as mentioned above, is determined as follows: The .007 true position tolerance of Fig. 27 would normally be based on the MMC size of the hole (.247). (Remember, MMC of the hole is a minimum-size hole or .250 – .003 = .247.) As the hole size deviates from the MMC size, the position of the hole is permitted to shift off its "true position," beyond the original tolerance zone, to the extent of that departure. The "bonus tolerance" of .013 illustrates the possible true-position tolerance should the hole be produced, for example, to the high-limit size of .253.

ASSIGNMENT:

1. Explain the true-position "bonus" tolerance.
2. How does the rectangular coordinate system differ from the true-position system?
3. What is the MMC of a hole .500 ± .005 dia?
4. Convert a ± .005 tolerance to a true-position tolerance.
5. Convert a ± .007 tolerance to a true-position tolerance.

TRUE POSITION

MATING PARTS - FLOATING FASTENER

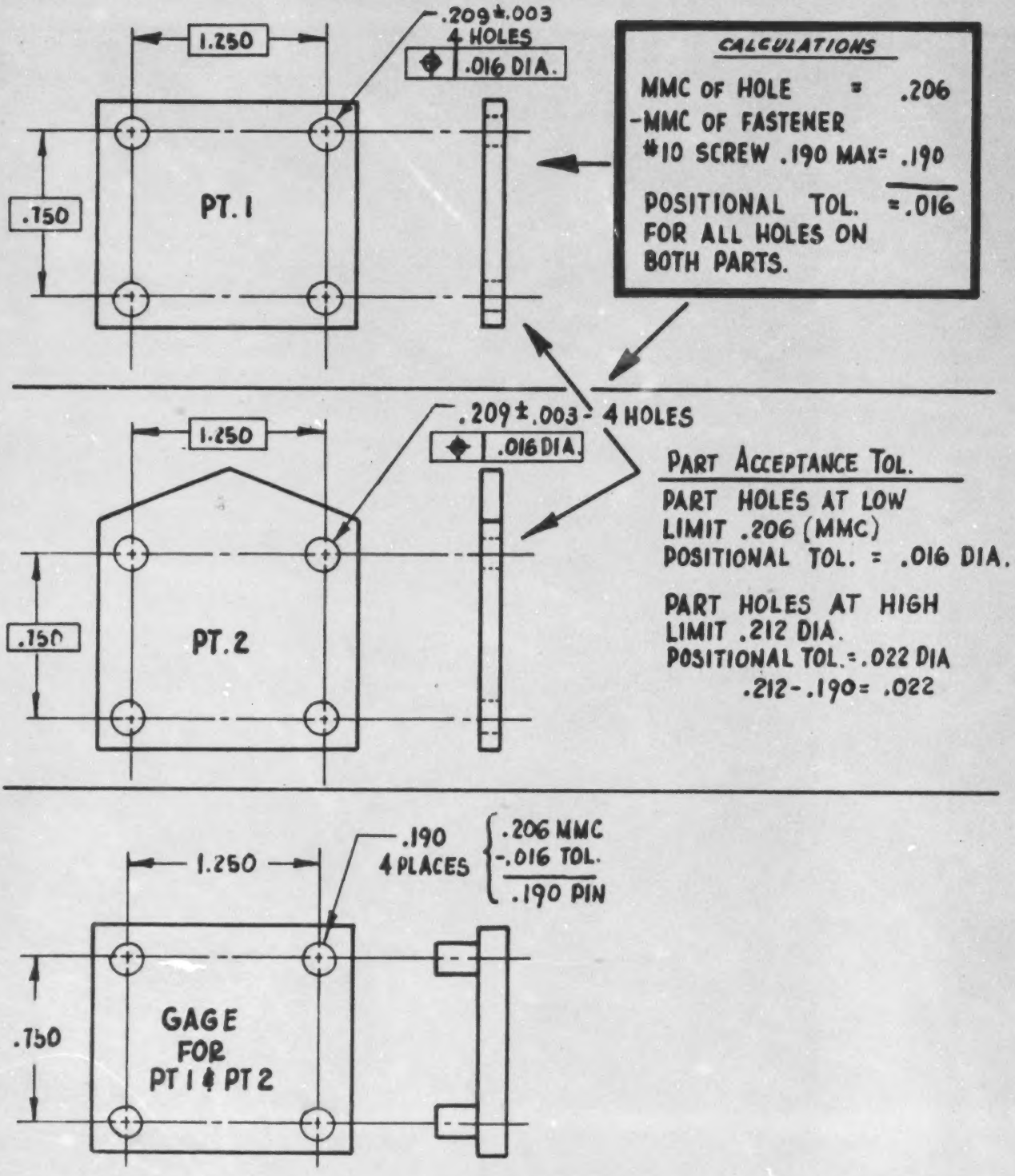


Figure 28

UNIT III
TOLERANCES OF POSITION

Lesson 3 — **Mating Parts: Floating-Fastener Condition**

Objectives: You will be able to identify and specify the floating-fastener condition and calculate the tolerances required.

Related Information:

1.0 An important function of layouts and drawings is to insure the fit between mating parts and mating assemblies. True-position dimensioning is especially useful in mating-parts situations. One of the most common of these situations is the floating fastener. You probably know this as two parts joined together with a machine screw and nut, both parts having clearance holes.

2.0 Fig. 28 shows two mating parts which are to be assembled with four machine screws (floating fasteners). With this type of design, the main concern is that the holes line up well enough so that the four screws will pass through both parts. Since the four screws are separate components, they are considered to have some "float" with respect to each other.

2.1 As you know, different sizes of machine screws require different sized clearance holes, and these holes are usually set by engineering standards.

3.0 The mating-part floating-fastener-condition method for determining tolerances may be calculated and specified by four different methods, which will be referred to as cases.

3.1 Case I: Balance Tolerance Application (shown in Fig. 28)

The calculations for our example are shown in the upper right-hand corner of the figure 28. Note that the same basic dimensions and positional tolerances are used on both parts. They are, of course, separate parts and are on separate drawings.

Case II: Optional Tolerance Distribution

If in the same example we desired to apply only .010 tolerance instead of .016 to part No. 1, the remaining portion of the tolerance not used – which is .006 – may be added to the tolerance of part No. 2. Part 2 tolerance becomes + .006 + .016 = .022.

(If specified on part 1) → ⌖ .010 DIA ⌖ .022 DIA ← (Specified on part 2).

	Part 1		Part 2
Original calculated tolerance	.016		.016
If tolerance assigned as above (−)	.010	(+)	.006
Tolerance unused	.006		.022

Case III: Hole sizes specified as different sizes – Averaging Method.

The clearance holes specified in Case I and Case II have the same size (dia.). Where they are specified as different sizes, the total true-position tolerance is equal to the *average* clearance between mating holes and fasteners. An example is shown below.

One part		Other part	
MMC hole	.200	MMC hole	.206
MMC fastener	−.190	MMC fastener	−.190
	.010		.016

$$\frac{.010 + .016}{2} = \frac{.026}{2} = .013$$

⌖ .013 DIA (specified on both parts)

Case IV – Hole sizes specified as different sizes – Individual Method.

Each part can be calculated separately for the allowable positional displacement, based on the difference between the MMC of the hole and the fastener. If one part, for example, had been .203 ± .003 – 4 holes, and the other part .209 ± .003 – 4 holes, then the method below would be used.

One part	
MMC hole	.200
MMC fastener	−.190
	.010

⌖ .010 DIA

Other part	
MMC hole	.206
MMC fastener	−.190
	.016

⌖ .016 DIA

4.0 Formulas used as a basis for the true-position floating-fastener calculation are:

4.1 To calculate true-position tolerance with fastener- and hole-size known

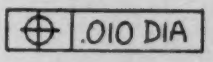
$$T = H - F$$

T = tolerance, H = MMC hole, and F = MMC fastener

Example:

Hole = .203 ± .003

Fastener = #10 screw – .190 MMC

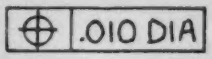
T = .200 – .190 = .010 or 

specified on part

4.2 Where the hole size or fastener size is to be derived from an established true-position tolerance, the formula is altered to:

$$H = T + F$$

Example of above:

Tolerance = 

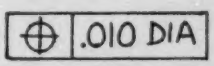
Fastener = #10 screw – .190 MMC

H = .010 + .190 = .200 dia hole at MMC

4.3 Another alteration of the formula can be:

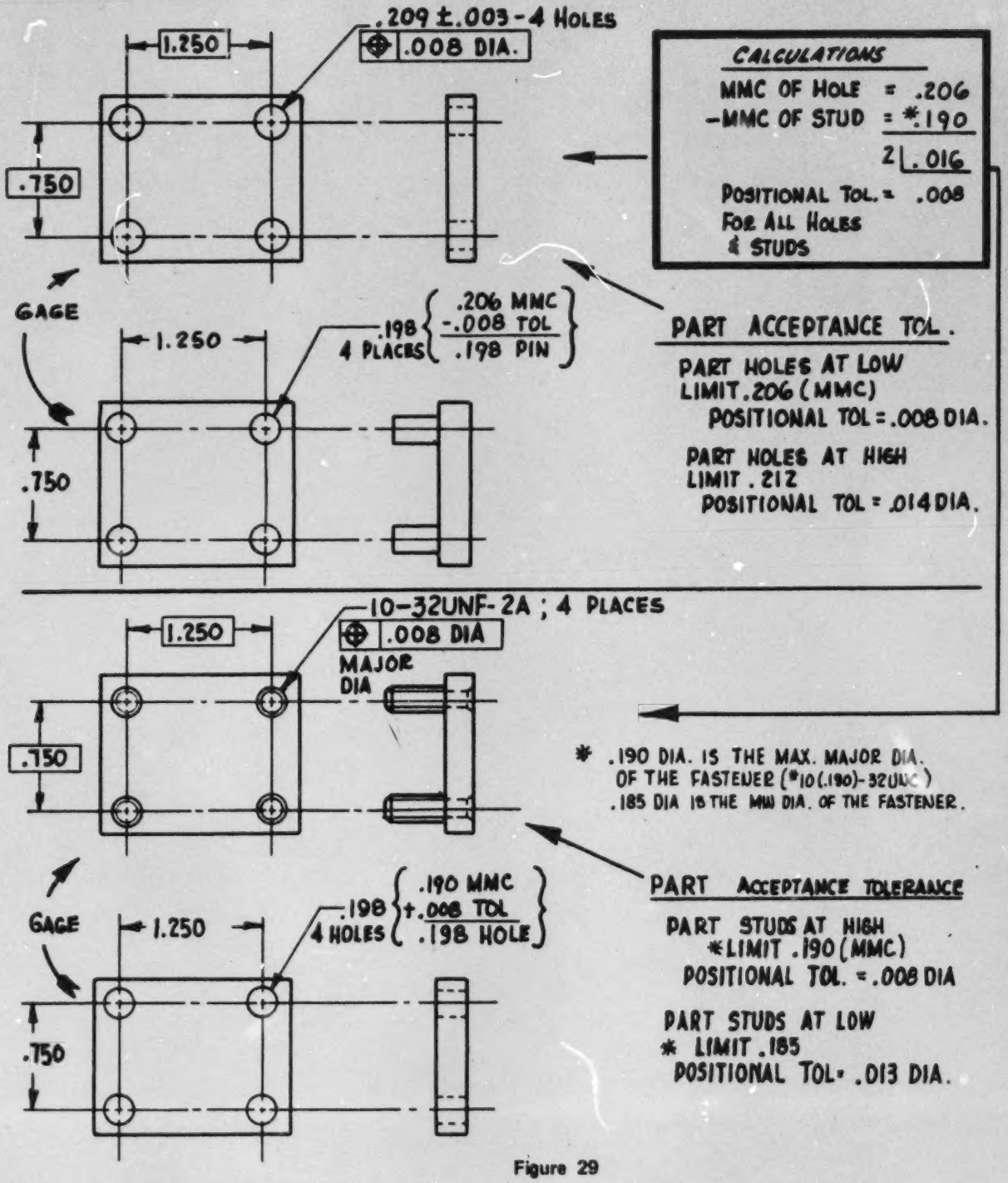
$$F = H - T$$

Hole = .203 ± .003 dia at MMC

Tolerance = .010 or 

F = .200 – .010 = .190 dia screw (#10) at MMC

TRUE POSITION
MATING PARTS – FIXED FASTENER



CALCULATIONS

MMC OF HOLE = .206
−MMC OF STUD = *.190
2 | .016
POSITIONAL TOL. = .008
FOR ALL HOLES & STUDS

PART ACCEPTANCE TOL.

PART HOLES AT LOW LIMIT .206 (MMC)
POSITIONAL TOL = .008 DIA.

PART HOLES AT HIGH LIMIT .212
POSITIONAL TOL = .014 DIA.

* .190 DIA. IS THE MAX. MAJOR DIA. OF THE FASTENER (*10(.190)-32UNF)
.185 DIA IS THE MIN DIA. OF THE FASTENER.

PART ACCEPTANCE TOLERANCE

PART STUDS AT HIGH
*LIMIT .190 (MMC)
POSITIONAL TOL. = .008 DIA

PART STUDS AT LOW
* LIMIT .185
POSITIONAL TOL = .013 DIA.

Figure 29

UNIT III
TOLERANCES OF POSITION

Lesson 4 — **Mating Parts: Fixed-Fastener Condition**

Objectives: You will be able to identify and specify the fixed-fastener condition and calculate the tolerances required.

Related Information:

1.0 When one of two mating parts has fixed features, such as threaded studs, i.e., studs (machine screws) which have been secured to a member by welding or staking, the term "fixed fastener" is used in describing this application.

2.0 The advantages of the MMC principle, discussed in the previous lesson, also apply here. However, with a fixed-fastener application, the difference between the MMC sizes of mating features must be divided between the two features, since the total true-position tolerance must be shared by the two mating features. In Fig. 29, the two mating features are the studs and the clearance holes. The four studs must fit through the four holes at assembly. (Note that the gage for each part is shown directly below that part.)

2.1 The true-position tolerances are determined by the mating features as they relate to each other at assembly. When one feature (the hole) is to be assembled within another on the basis of MMC sizes and "worst" condition at assembly, the clearance, or total tolerance, must be divided for assignment to *each* of the mating-part features. In the illustration, the .016 is divided equally, with .008 diameter true-position tolerance assigned to each mating-part feature (stud and hole). The total tolerance of .016 can be distributed to the two parts as desired, so long as the total is .016 (e.g., .010 + .006 = .016; .011 + .005 = .016, etc.).

2.2 Any deviation of part features in size from MMC will increase the calculated true position by an amount equal to the total deviation. The upper-part hole tolerance could increase to .014 (.008 + .006 = .014); the lower part to a possible .013. Tolerances must be established before the *amount of increase in tolerance* can be determined.

3.0 Gages can be used to determine the fit of mating parts, and the use of gages in large production runs is almost essential. Therefore, a study to determine how gage pins are designed is in order. A gage of the type we are talking about is a functional gage. Functional gages are designed to simulate mating parts at their worst condition.

3.1 The gage pins of the upper part are determined by the MMC hole size minus the true-position tolerance.

3.2 The gage holes on the lower part are determined by the MMC (O.D.) size of the 10-32 pins plus the stated true-position tolerance.

4.0 The calculations on these parts illustrates a balanced tolerance application in which the total permissible true-position tolerance is equally divided, .008 on each part. As indicated in 2.1 above, however, the total true-position tolerance can be distributed as desired.

ASSIGNMENT:

1. What fasteners other than studs are classified as fixed fasteners?
2. Which fastener condition (fixed or floating) requires tighter tolerances?
3. Calculate gage-pin true-position tolerance if the hole dimension is .125 ± .003 dia. and ⌖ | .004 DIA.
4. Calculate the true-position tolerance of .136 ± .005 dia hole and a #4-40 MMC stud which has a .112 dia.

UNIT III
TOLERANCES OF POSITION

Lesson 5 — **Location of Holes by Angular Dimensions**

Objective: You will be able to calculate the tolerances required for fixed fasteners located by angular dimensions.

Related Information:

1.0 The illustration in Fig. 30 shows true-position tolerancing applied to two mating parts with a round-hole pattern. The same reasoning applies here as in the previous examples except that the basic dimensions are angles (45° angles, 8 places) and a diameter (the 1.250 dia.).

1.1 To determine the positional tolerances for each part, the MMC of the hole and the MMC of the stud are first used to determine the *total* positional tolerance. This is then divided by 2 to give the positional tolerance value for each part.

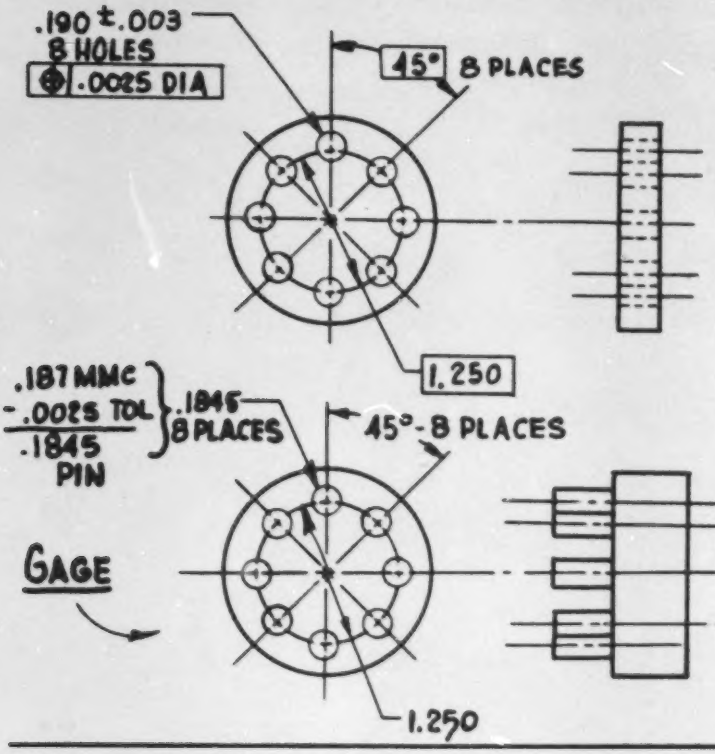
1.2 Note again how the positional tolerance increases as the holes in the upper part and the studs in the lower part depart from their MMC sizes, that is, when the holes get larger and the pins get smaller during production (as they would normally do, due to machine wear, etc.).

1.3 The calculations illustrate a balanced tolerance application, in which the total permissible true-position tolerance of the two parts is equally divided (in the example, .0025 on each part). Again, the total true-position tolerance can be distributed as desired: for example, .002 on one part, .003 on the other, etc, so long as they total the tolerance calculated (in this case, .005).

ASSIGNMENT:

1. In the example in Fig. 30, calculate the true-position tolerance when the holes are .261 ± .005 dia and the studs are .250 MMC.

TRUE POSITION
MATING PARTS - FIXED FASTENER



CALCULATIONS

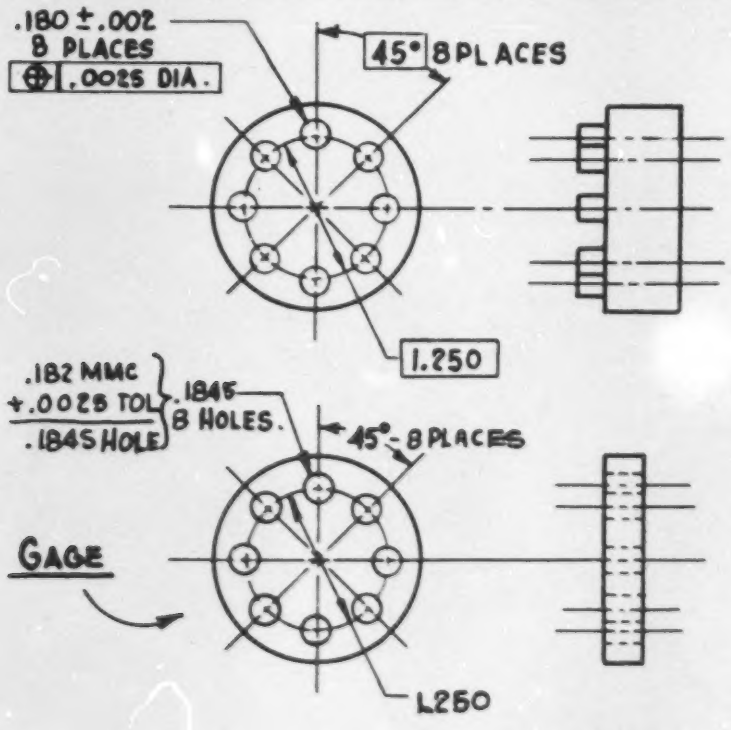
MMC OF HOLE = .187
- MMC OF STUD = .182
2 | .005

POSITIONAL TOL. FOR ALL HOLES & STUDS. = .0025

PART ACCEPTANCE TOL.

PART HOLES AT LOW LIMIT .187 (MMC)
POSITIONAL TOL = .0025

PART HOLES AT HIGH LIMIT .193
POSITIONAL TOL. = .0085



PART ACCEPTANCE TOL.

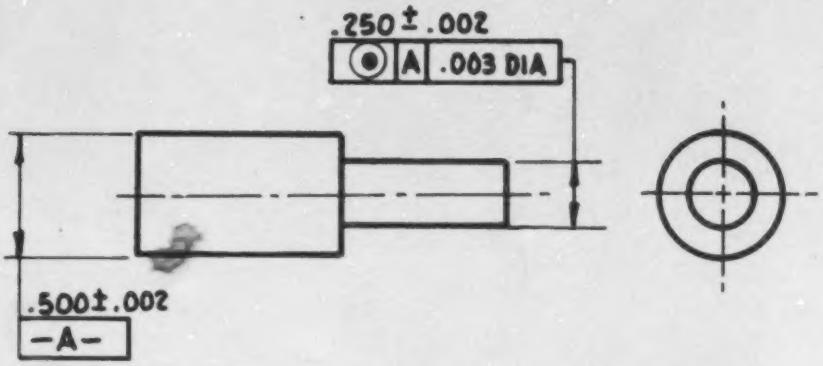
PART STUDS AT HIGH LIMIT .182 (MMC)
POSITIONAL TOL. = .0025

PART STUDS AT LOW LIMIT .178
POSITIONAL TOL. = .0065

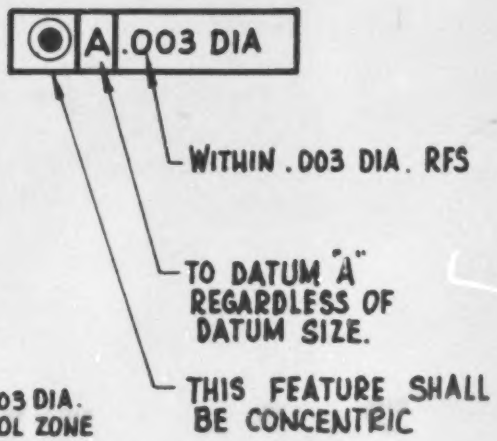
Figure 30

CONCENTRICITY - ◉

CONCENTRICITY TOLERANCE:
CONCENTRICITY TOL. IS THE DIAMETER OF THE CYLINDRICAL TOL. ZONE WITHIN WHICH THE AXIS OF THE FEATURES MUST LIE, THE AXIS OF THE TOL. ZONE COINCIDING WITH THE AXIS OF THE DATUM FEATURES.



INTERPRETATION



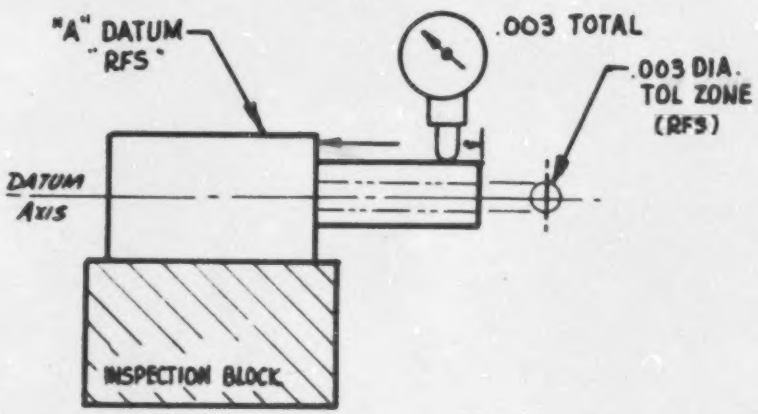


Figure 31

UNIT III
TOLERANCES OF POSITION

Lesson 6 — **Concentricity**

Objective: You will be able to identify and specify concentricity.

Related Information:

1.0 Definition: Concentricity is a condition in which two or more features (cylinders, cones, spheres, hexagons, etc.) in any combination have a common axis.

Concentricity tolerance: Concentricity tolerance is the diameter of the cyclindrical tolerance zone within which the axes of the features must lie; the axis of the tolerance zone must concide with the axes of the datum features.

1.1 Concentricity is a condition where two or more features on a part have a *common axis.* The concentricity tolerance zone is the diameter of the cylindrical tolerance zone within which the axes of the features must lie.

1.2 In the illustration in Fig. 31, the datum diameter is .500 ± .002, and the .250 ± .002 diameter is the diameter which must be concentric with the .500 diameter within .003 total.

The .003 dia tolerance zone is the amount of deviation of the .250 dia axis from the .500 dia axis as measured with a dial indicator gage.

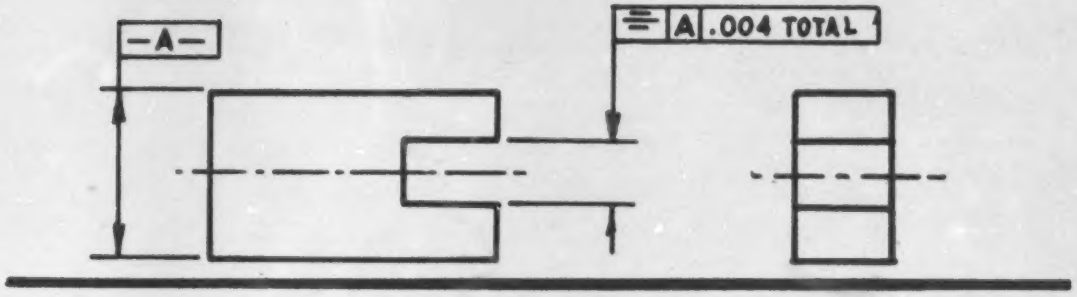
1.3 Concentricity is a type of positional tolerancing. It always involves two or more basically coaxial features of size, and controls *the amount by which the axes* of the features *fail* to coincide. Concentricity is always used on an RFS basis.

2.0 Where interrelated features are basically coaxial (such as the cylinders in illustration 31), the draftsman should consider using types of tolerances that are more economical to test, for example, true position or runout, before considering concentricity. The cost of inspection is high to measure concentricity, and other methods (e.g., runout) are more commonly used in industry today.

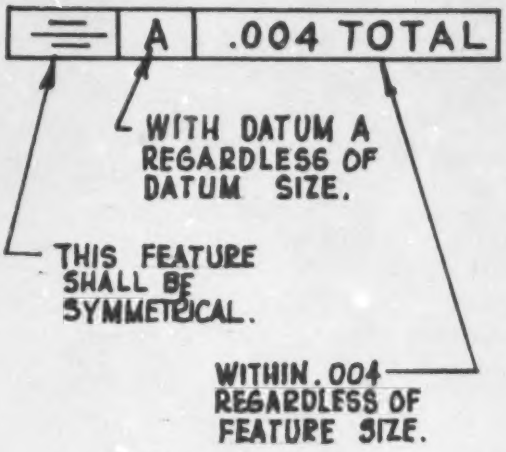
ASSIGNMENT:

1. Define "concentricity" in your own words.
2. Sketch an application of concentricity. Show all dimensions and tolerances.
3. Define "coaxial features."
4. Using the same sketch you made for question 2, apply an equivalent runout concept to your sketch.
5. Can you think of a condition where concentricity tolerance should definitely be used despite the extra cost involved?

SYMMETRY ⌯



INTERPRETATION


⌯ A .004 TOTAL
WITH DATUM A REGARDLESS OF DATUM SIZE.
THIS FEATURE SHALL BE SYMMETRICAL.
WITHIN .004 REGARDLESS OF FEATURE SIZE.

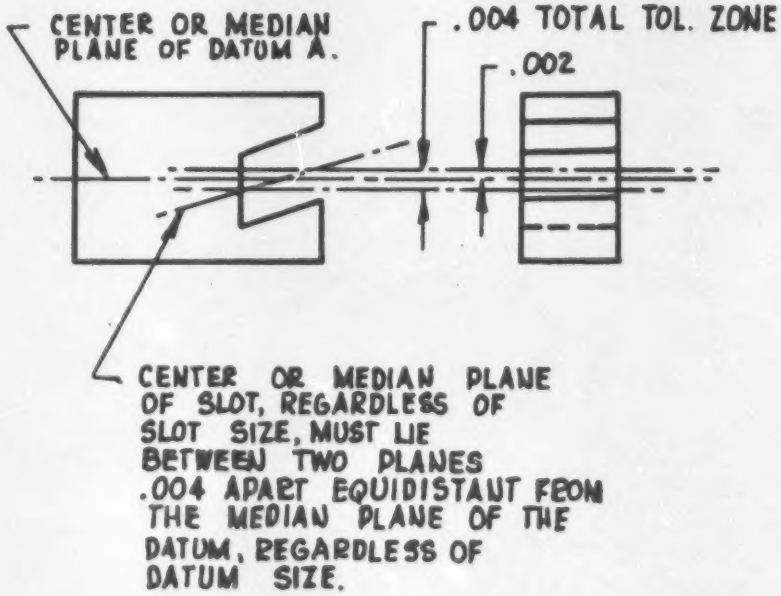

CENTER OR MEDIAN PLANE OF DATUM A.
.004 TOTAL TOL. ZONE
.002
CENTER OR MEDIAN PLANE OF SLOT, REGARDLESS OF SLOT SIZE, MUST LIE BETWEEN TWO PLANES .004 APART EQUIDISTANT FROM THE MEDIAN PLANE OF THE DATUM, REGARDLESS OF DATUM SIZE.

Figure 32

UNIT III
TOLERANCES OF POSITION

Lesson 7 **Symmetry**

Objective: You will be able to identify and specify symmetry.

Related Information:

1.0 Definition: **Symmetry** is a condition in which a part or feature has the same contour on opposite sides of a median plane, or in which a feature has a common plane with a datum plane. The **median plane** is the middle, or center, plane of a feature.

1.1 **Symmetry tolerance:** The tolerance governing the symmetry of a feature with respect to a datum is the distance between two parallel planes between which the median plane (center plane) of the feature must lie, the parallel planes being parallel to, and equally distant from, the median plane, or axis, of the datum feature.

2.0 Symmetry deals with the position of actual features with respect to established center planes or axes. The illustration in Fig. 32 shows a part using symmetry tolerancing on the slot; the part width is established as the datum. Both the slot and the outside edges of the part established by datum A have a common center-reference plane. This reference plane is the center, or median plane of the datum feature. Then the center, or median, plane of the toleranced feature is compared with these two median planes. The result of this comparison shows whether or not the symmetry tolerance has been met.

2.1 The figure interpretation shows the relationship and establishment of the symmetry tolerance zone with respect to the datum center plane. Note that the center plane, or median plane, of datum A establishes the mid-plane of the .004 tolerance zone.

2.2 The median plane of the produced slot, regardless of the slot size, must lie between these two planes, which are .004 apart and equidistant (or .002) from the median plane of the datum, regardless of datum size. Note that the median plane of the slot, as determined by the actual slot, may fall at an angle.

3.0 True-position characteristics should be used instead of symmetry whenever possible. Symmetry is, however, a valid characteristic and may be applied if desired, but on an *RFS basis only*.

ASSIGNMENT:

1. Define the term "symmetry" in your own words.
2. What is a symmetry tolerance?
3. What is a median plane?
4. Can symmetry be used with an MMC condition?
5. Sketch an application of symmetry, showing all dimensions and tolerances.
6. When might symmetry possibly be used for tolerancing rather than true-position characteristics?

APPENDIX

CONVERSION OF TRUE POSITION (CYLINDRICAL) TOL ZONES TO/FROM COORDINATE TOLERANCE ZONES

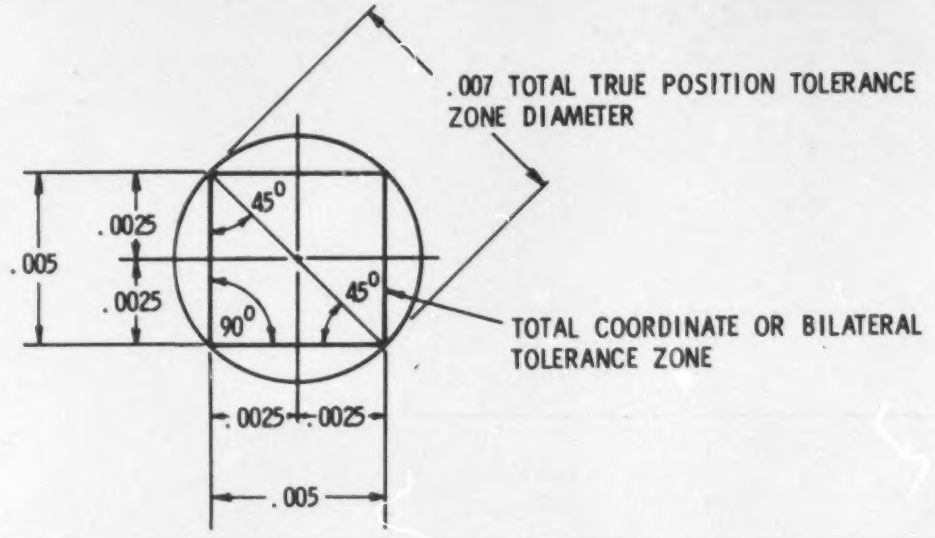
⌖ TO ±

TOTAL TP TOL ZONE X .7071I = TOTAL COORDINATE TOLERANCE ZONE

EXAMPLE: .007 TP TOL X .7071I = .00495 } .005 TOTAL COORDINATE TOL OR .0025 BILATERAL TOL

RULE OF THUMB:

USE .7 (OR 70%) OF TOTAL TP TOL ZONE TO CONVERT QUICKLY IN NON-CRITICAL APPLICATIONS, e.g. .7 X .007 = .0049 OR .005 (±0025)



± TO ⌖

TOTAL COORDINATE TOL ZONE X 1.4142 = TOTAL TP TOL ZONE

EXAMPLE: .005 TOTAL COORDINATE TOL OR .0025 BILATERAL TOL X 2 } X 1.4142 = .007 TOTAL TP TOL

RULE OF THUMB:

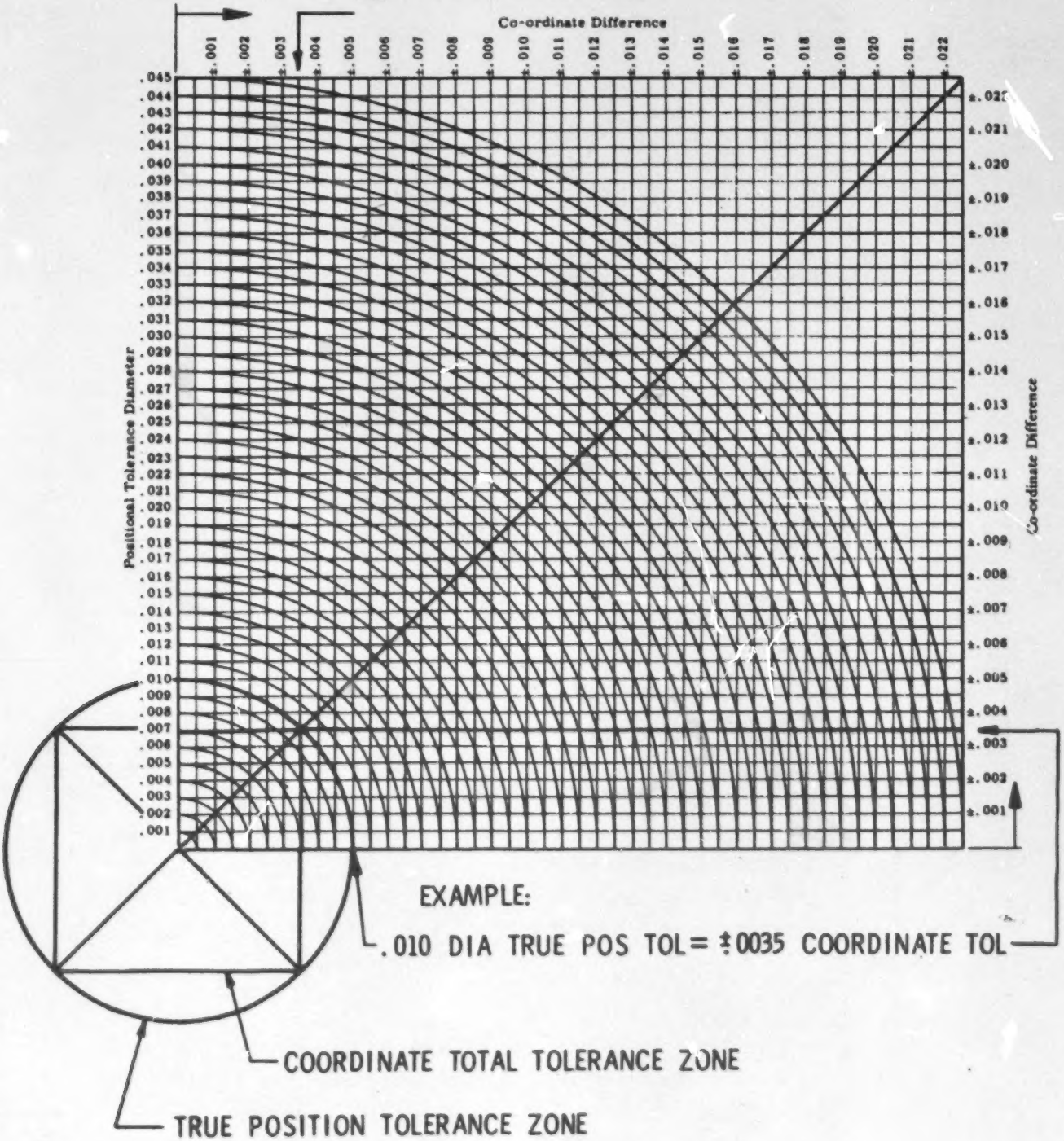
USE 1.4 TIMES TOTAL COORD TOL ZONE TO CONVERT QUICKLY IN NON-CRITICAL APPLICATIONS, e.g. 1.4 X .005 = .007 TP TOL

Reference: Conversion chart from Sandia Corporation Process Standard 9900011.

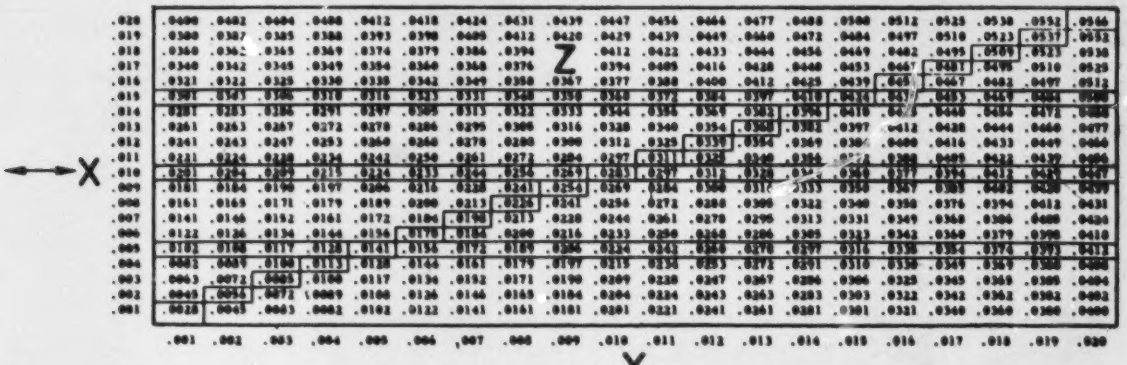
CONVERSION CHART:

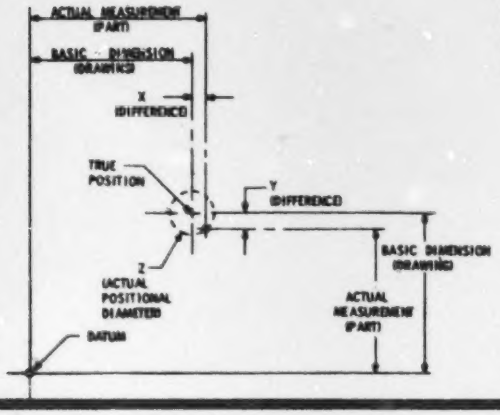
TRUE POSITION TOL TO COORDINATE TOL

COORDINATE TOL TO TRUE POSITION TOL



CONVERSION OF COORDINATE MEASUREMENTS TO TRUE POSITION LOCATION

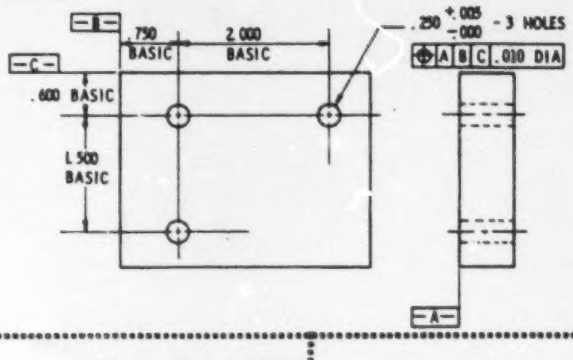




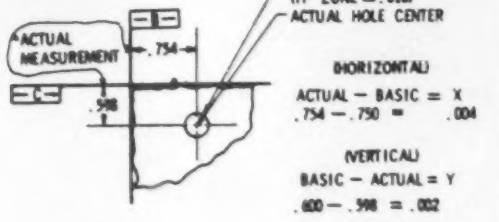
FORMULA

$$Z = 2\sqrt{X^2 + Y^2}$$

EXAMPLE



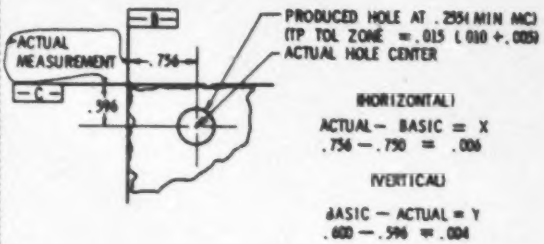
CONVERSION



(HORIZONTAL)
ACTUAL − BASIC = X
.754 − .750 = .004

(VERTICAL)
BASIC − ACTUAL = Y
.600 − .598 = .002

FROM CHART ABOVE .004 (X) AND .002 (Y) GIVE .0089 DIA. HOLE LOCATION IS WITHIN SPECIFIED .010 DIA. HOLE LOCATION ACCEPTABLE



(HORIZONTAL)
ACTUAL − BASIC = X
.756 − .750 = .006

(VERTICAL)
BASIC − ACTUAL = Y
.600 − .596 = .004

FROM CHART ABOVE .006 (X) AND .004 (Y) GIVE .0144 DIA. HOLE LOCATION IS WITHIN PERMISSIBLE .015 DIA WITH HOLE AT HIGH LIMIT. HOLE LOCATION ACCEPTABLE.

Reference: Conversion Chart from Sandia Corporation Process Standard 9900011.

COMPARISON BETWEEN MIL-STD-8C & USASI Y14.5 - 1966

MAJOR DIFFERENCES

MIL-STD-8C

SYMBOLS

GEOMETRIC CHARACTERISTICS

Symbol	Characteristics
—	FLATNESS OR STRAIGHTNESS ♦
∠	ANGULARITY
⊥	PERPENDICULARITY (SQUARENESS)
∥	PARALLELISM
○	ROUNDNESS (CIRCULARITY)
(none)	CONTOUR
⌖	TRUE POSITION
◎	CONCENTRICITY
⌯	SYMMETRY

MODIFIERS

Ⓜ MAXIMUM MATERIAL CONDITION
Ⓢ REGARDLESS OF FEATURE SIZE

(♦) Symbol Formerly ⌒ under MIL-STD-8B.

TOLERANCES

All tolerances specified are totals (See also below)

DESCRIPTION OF TOLERANCE ZONE SHAPE

Must be specified for:

True position of holes or other cylindrical features,

DIA is preferred ⌖ .010 DIA

R may be specified ⌖ .005 R

True position of slots or tabs, TOTAL width is preferred

⌖ .010 TOTAL

EITHER SIDE may be specified

⌖ .005 EITHER SIDE

Must be specified for:

Roundness

Either tolerance on the radius or on the diameter may be specified.

○ .005 ON R
or
○ .010 ON DIA

MISC.

Vertical lines required in feature control symbol to separate characteristics on Straightness / Flatness and Roundness

— | .XXX

○ | .XXX

USA Y14.5 - 1965

SYMBOLS & CHARACTERISTICS (SEE ALSO TEXT)

Added to those used in MIL-STD-8C:

Cylindricity ⌭
Runout ↗
Profile of Any Surface (Contour) ⌓ (Symbol only added)
Profile of Any Line ⌒

Changed:

Flatness ▱

MODIFIERS

Same as MIL-STD-8C

TOLERANCES

All tolerances specified are totals (See also below)

DESCRIPTION OF TOLERANCE ZONE SHAPE

Must be specified for:

True position of holes or other cylindrical features,

DIA is preferred ⌖ .010 DIA

R may be specified ⌖ .005 R

True position of slots or tabs,

TOTAL width is preferred ⌖ .010 TOTAL

Half width may be specified with R ⌖ .010 R

Concentricity of features (RFS only, if MMC use ⌖) ◎ .003 DIA

Symmetry of features (RFS only, if MMC use ⌖) ⌯ .003 TOTAL

MISC.

Datum Precedence Expansion
Three Plane Concept
Datum Targets
Vertical lines required in all feature control symbols to separate characteristics ∠ .005

Multiple Datum features establishing a single datum reference.

↗ | X-Y | .002

Basic dimensions may be boxed in as optional symbolic method.

.560

Double datum letter references, e.g., AA, AB, etc. may be used where single letter alphabet exhausted.

COMPARISON MIL-STD-8C, USASI Y14.5, ISO

BASIC, TOLERANCE, DATUMS, MISC.

	MIL-STD-8C	USASI Y14.5	ISO
TOLERANCE	TOTAL TOLERANCE SPECIFIED (SEE "SHAPE OF TOL ZONE" BELOW)	TOTAL TOLERANCE SPECIFIED (SEE "SHAPE OF TOL ZONE" BELOW)	TOTAL VALUE SPECIFIED
SHAPE OF TOLERANCE ZONE	SPECIFIED FOR TRUE POSITION, ROUNDNESS, CONCENTRICITY.	SPECIFIED FOR TRUE POSITION CONCENTRICITY SYMMETRY	ZONE IS WIDTH IN DIRECTION OF LEADER ARROW. SPECIFIED WHERE ZONE IS CYLINDRICAL ⌖ Ø 0,4 Ⓜ
SEQUENCE OF GEOMETRIC CHARACTERISTIC, DATUM, & TOLERANCE WITHIN FEATURE CONTROL SYMBOL	⌖ A B .005 DIA	⌖ A B .005 DIA	⌖ Ø 0,4 Ⓜ A B
BASIC DIMENSION	.505 BASIC .505 BSC NOTATION	[.505] .505 BASIC .505 BSC NOTATION	[127]

COMPARISON MIL-STD-8C, USASI Y14.5, ISO SYMBOLS

CHARACTERISTIC	MIL-STD-8C	USASI Y14.5	ISO
STRAIGHTNESS	—	—	—
FLATNESS	—	▱	▱
ANGULARITY	∠	∠	∠
PERPENDICULARITY (SQUARENESS)	⊥	⊥	⊥
PARALLELISM	||	|| or //	//
CONCENTRICITY	◉	◉ or ◎	◎
TRUE POSITION	⌖	⌖	⌖
ROUNDNESS (CIRCULARITY)	○	○	○
SYMMETRY	⌯	⌯	⌯
PROFILE OF ANY LINE	NONE	⌒	⌒
PROFILE OF ANY SURFACE (CONTOUR)	USED NOTE	⌓	⌓
RUNOUT	USED RFS CONCENTRICITY	↗	↗
CYLINDRICITY	NONE	⌭	⌭
DATUM FEATURE	-A-	-A-	A
MAXIMUM MATERIAL CONDITION (MMC)	Ⓜ	Ⓜ	Ⓜ
REGARDLESS OF FEATURE SIZE (RFS)	Ⓢ	Ⓢ	NONE (ASSUMED UNLESS SPECIFIED MMC)